

CVNC™ -T2 User Guide

CADD5® 5 Revision 6.1.1

DOC36795-004

Copyright © 2001 Parametric Technology Corporation. All Rights Reserved.

User documentation from Parametric Technology Corporation (PTC) is subject to copyright laws of the United States and other countries and is provided under a license agreement, which restricts copying, disclosure, and use of such documentation. PTC hereby grants to the licensed user the right to make copies in printed form of PTC user documentation provided on software or documentation media, but only for internal, noncommercial use by the licensed user in accordance with the license agreement under which the applicable software and documentation are licensed. Any copy made hereunder shall include the Parametric Technology Corporation copyright notice and any other proprietary notice provided by PTC. User documentation may not be disclosed, transferred, or modified without the prior written consent of PTC and no authorization is granted to make copies for such purposes.

Information described in this document is furnished for general information only, is subject to change without notice, and should not be construed as a warranty or commitment by PTC. PTC assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

The software described in this document is provided under written license agreement, contains valuable trade secrets and proprietary information, and is protected by the copyright laws of the United States and other countries. UNAUTHORIZED USE OF SOFTWARE OR ITS DOCUMENTATION CAN RESULT IN CIVIL DAMAGES AND CRIMINAL PROSECUTION.

Registered Trademarks of Parametric Technology Corporation or a Subsidiary

Advanced Surface Design, CADD5, CADDShade, Computervision, Computervision Services, Electronic Product Definition, EPD, HARNESSDESIGN, Info*Engine, InPart, MEDUSA, Optegra, Parametric Technology, Parametric Technology Corporation, Pro/ENGINEER, Pro/HELP, Pro/INTRALINK, Pro/MECHANICA, Pro/TOOLKIT, PTC, PT/Products, Windchill, InPart logo, and PTC logo.

Trademarks of Parametric Technology Corporation or a Subsidiary

3DPAINT, Associative Topology Bus, Behavioral Modeler, BOMBOT, CDRS, CounterPart, CV, CVact, CVaec, CVdesign, CV-DORS, CVMAC, CVNC, CVToolmaker, DesignSuite, DIMENSION III, DIVISION, DVSAFEWORK, DVS, e-Series, EDE, e/ENGINEER, Electrical Design Entry, Expert Machinist, Expert Toolmaker, Flexible Engineering, i-Series, ICEM, Import Data Doctor, Information for Innovation, ISSM, MEDEA, ModelCHECK, NC Builder, Nitidus, PARTBOT, PartSpeak, Pro/ANIMATE, Pro/ASSEMBLY, Pro/CABLING, Pro/CASTING, Pro/CDT, Pro/CMM, Pro/COMPOSITE, Pro/CONVERT, Pro/DATA for PDGS, Pro/DESIGNER, Pro/DESKTOP, Pro/DETAIL, Pro/DIAGRAM, Pro/DIEFACE, Pro/DRAW, Pro/ECAD, Pro/ENGINE, Pro/FEATURE, Pro/FEM-POST, Pro/FLY-THROUGH, Pro/HARNESS-MFG, Pro/INTERFACE for CADD5, Pro/INTERFACE for CATIA, Pro/LANGUAGE, Pro/LEGACY, Pro/LIBRARYACCESS, Pro/MESH, Pro/Model.View, Pro/MOLDESIGN, Pro/NC-ADVANCED, Pro/NC-CHECK, Pro/NC-MILL, Pro/NC-SHEETMETAL, Pro/NC-TURN, Pro/NC-WEDM, Pro/NC-Wire EDM, Pro/NCPOST, Pro/NETWORK ANIMATOR, Pro/NOTEBOOK, Pro/PDM, Pro/PHOTORENDER, Pro/PHOTORENDER TEXTURE LIBRARY, Pro/PIPING, Pro/PLASTIC ADVISOR, Pro/PLOT, Pro/POWER DESIGN, Pro/PROCESS, Pro/REPORT, Pro/REVIEW, Pro/SCAN-TOOLS, Pro/SHEETMETAL, Pro/SURFACE, Pro/VERIFY, Pro/Web.Link, Pro/Web.Publish, Pro/WELDING, Product Structure Navigator, PTC i-Series, Shaping Innovation, Shrinkwrap, The Product Development Company, Virtual Design Environment, Windchill e-Series, CV-Computervision logo, DIVISION logo, and ICEM logo.

Third-Party Trademarks

Oracle is a registered trademark of Oracle Corporation. Windows and Windows NT are registered trademarks of Microsoft Corporation. Java and all Java based marks are trademarks or registered trademarks of Sun Microsystems, Inc. CATIA is a registered trademark of Dassault Systems. PDGS is a registered trademark of Ford Motor Company. SAP and R/3 are registered trademarks of SAP AG Germany. FLEX/m is a registered trademark of GLOBEtrouter Software, Inc. VisTools library is copyrighted software of Visual Kinematics, Inc. (VKI) containing confidential trade secret information belonging to VKI. HOOPS graphics system is a proprietary software product of, and copyrighted by, Tech Soft America, Inc. All other brand or product names are trademarks or registered trademarks of their respective holders.

UNITED STATES GOVERNMENT RESTRICTED RIGHTS LEGEND

This document and the software described herein are Commercial Computer Documentation and Software, pursuant to FAR 12.212(a)-(b) or DFARS 227.7202-1(a) and 227.7202-3(a), and are provided to the Government under a limited commercial license only. For procurements predating the above clauses, use, duplication, or disclosure by the Government is subject to the restrictions set forth in subparagraph (c)(1)(ii) of the Rights in Technical Data and Computer Software Clause at DFARS 252.227-7013 or Commercial Computer Software-Restricted Rights at FAR 52.227-19, as applicable.

Parametric Technology Corporation, 140 Kendrick Street, Needham, MA 02494-2714

8 January 2001

Table of Contents

Preface

Related Documents	ix
Book Conventions	x
Window Managers and the User Interface	xi
Online User Documentation	xi
Online Command Help	xii
Printing Documentation	xii
Resources and Services	xiii
Documentation Comments	xiii

Introduction

About CVNC-T2	1-2
Product Description	1-2
Job Control File	1-4
What Is a JCF?	1-4
JCF Example	1-4
How Are JCFs Created?	1-5
CVNC-T2 Features	1-6
System Variables	1-6
System Monitoring Tools	1-6
Macros	1-6
Command Files	1-7
Output	1-7
Output Pass-through Statements	1-7

Modifying the CVNC-T2 Language Environment _____	1-8
--	-----

Using CVNC-T2

Entering CVNC-T2 _____	2-2
Entering and Editing Data _____	2-4
Entering Coordinate and Entity Data _____	2-4
Editing JCF Data _____	2-5

Job Setup

Configuring the Machine _____	3-2
Using DATUM _____	3-2
Using HOMEPT FROM _____	3-3
Using CONFIG _____	3-5
Defining Tool Parameters _____	3-7

Operation Setup

Selecting a Tool and Changing the Active Cplane _____	4-2
Selecting a Tool _____	4-2
Changing the Active Cplane _____	4-2
Specifying Feed Rate and Spindle Speed _____	4-5
Speed Rate _____	4-5
Feed Rate _____	4-5
Modal Feed Rates _____	4-5
Setting STOCK Offsets _____	4-7
STOCK and Fillets _____	4-10
Modifiers Used with STOCK _____	4-10
Fillet: Floated or Not Floated _____	4-11
Conditions when a Fillet is Floated _____	4-11
Conditions when a Fillet Is Not Floated _____	4-12
Specifying Tolerance _____	4-13

Changing the Tool Gage Length _____	4-14
-------------------------------------	------

Motion

Motion Types _____	5-2
Rough Turning _____	5-3
Contoured and Noncontoured Approaches and Retractions ____	5-4
Specifying Maximum Depth for Turning Motion _____	5-6
Specifying Minimum Depth for Turning Motion _____	5-7
Undercutting During Rough Turning _____	5-8
Multiple Undercut Operation _____	5-10
Profiling _____	5-14
Threading _____	5-15
Thread Type _____	5-15
Multiple Thread Starts _____	5-16
Thread Location _____	5-17
Threading Passes _____	5-19
Threading Into and Out of Material and Gaging Passes _____	5-21
Threading Into and Out of Material _____	5-21
Gaging Passes _____	5-21
Hole Processing _____	5-23
Controlling Hole Depth _____	5-23
Controlling Clearance Distances _____	5-24
Identifying the Machining Location _____	5-25
Specifying Drilling Methods _____	5-25
DISPLAY CYCLE Command _____	5-27
Hole Processing for a Tapped Hole _____	5-28
Example _____	5-28
Additional Motion Commands _____	5-29

Preface

The *CVNC-T2 User Guide* provides instructions for CVNC-T2 commands and procedures.

It is written for NC part programmers, operators, and manufacturing engineers using CVNC to produce control tapes for numerically controlled machine tools. As a CVNC user, you should be familiar with the CADD5 and the CVNC programming environment.

Related Documents

The following documents may be helpful as you use the *CVNC-T2 User Guide*:

- *Understanding CVNC*
- *CVNC System User Guide and Menu Reference*
- *CVNC Editor Guide*
- *Customizing CVNC*
- *CVNC-T2 Command Reference*
- *CVNC Master Index*
- *CVNC System Variables Guide*
- *CVNC Work Examples*
- *Customizing CVNC*
- *CVMAC Language Reference*

Book Conventions

The following table illustrates and explains conventions used in writing about CADDs applications.

Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d ₁ or P1	Marks a location or entity selection in graphic examples.
User input in CADDs text fields and on any command line	<code>cvaec.hd.data.param</code> <code>tar -xvf /dev/rst0</code>	Enter the text in a CADDs text field or on any command line.
System output	<code>Binary transfer complete.</code>	Indicates system responses in the CADDs text window or on any command line.
Variable in user input	<code>tar -cvf /dev/rst0 filename</code>	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname with an actual tag name.
CADDs commands and modifiers	INSERT LINE TANTO	Shows CADDs commands and modifiers as they appear in the command line interface.
Text string	"SRFGROUPA" or 'SRFGROUPA'	Shows text strings. You must enclose text string with single or double quotation marks.
Integer	<i>n</i>	Supply an integer for the <i>n</i> .
Real number	<i>x</i>	Supply a real number for the <i>x</i> .
#	<code># mkdir /cdrom</code>	Indicates the root (superuser) prompt on command lines.
%	<code>% rlogin remote_system_name -l root</code>	Indicates the C shell prompt on command lines.
\$	<code>\$ rlogin remote_system_name -l root</code>	Indicates the Bourne shell prompt on command lines.

Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADDSS can change. Refer to the following table:

Look and Feel of User Interface Elements

User Interface Element	Common Desktop Environment (CDE) on Solaris, HP, DEC, and IBM	Window Manager Other Than CDE on Solaris, HP, DEC, IBM, SGI, and NT
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can view the online documentation in the following ways:

- From an HTML browser
- From the Information Access button on the CADDSS desktop or the Local Data Manager (LDM)

Please note: The LDM is valid only for standalone CADDSS.

You can also view the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,
 - `/usr/apl/caddss/data/html/htmlldoc/` (UNIX)
 - `Drive:\usr\apl\caddss\data\html\htmlldoc\` (Windows NT)
2. Click `mainmenu.html`. A list of available CADDSS documentation appears.
3. Click the book title you want to view.

From the Information Access Button on the CADDSS Desktop or LDM:

1. Start CADDSS.
2. Choose Information Access, the *i* button, in the top-left corner of the CADDSS desktop or the LDM.
3. Choose DOCUMENTATION. A list of available CADDSS documentation appears.
4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.
2. Point your browser to:
 - CDROM_mount_point/html/doc/mainmenu.html (UNIX)
 - CDROM_Drive:\html\doc\mainmenu.html (Windows NT)

Online Command Help

You can view the online command help directly from the CADDs desktop in the following ways:

- From the Information Access button on the CADDs desktop or the LDM
- From the command line

From the Information Access Button on the CADDs Desktop or LDM:

1. Start CADDs.
2. Choose Information Access, the *i* button, in the top-left corner of the CADDs desktop or the LDM.
3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb-noun combination as follows:

```
#01#!INSERT LINE
```

Printing Documentation

A PDF (Portable Document Format) file is included on the CD-ROM for each online book. See the first page of each online book for the document number referenced in the PDF file name. Check with your system administrator if you need more information.

You must have Acrobat Reader installed to view and print PDF files.

The default documentation directories are:

- /usr/apl/cadds/data/html/pdf/doc_number.pdf (UNIX)
- CDROM_Drive:\usr\apl\cadds\data\html\pdf\doc_number.pdf (Windows NT)

Resources and Services

For resources and services to help you with PTC (Parametric Technology Corporation) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback in the following ways:

- Send comments electronically to `doc-webhelp@ptc.com`.
- Fill out and mail the PTC Documentation Survey located in the *PTC Customer Service Guide*.

Introduction

This chapter provides an overview of CVNC-T2.

- About CVNC-T2
- Job Control File
- CVNC-T2 Features

About CVNC-T2

CVNC-T2 is an interactive, graphics software package that generates part programs for turning and hole processing lathe operations. CVNC-T2 includes a default language environment that consists of CVNC commands, macros, and a set of output pass-through statements.

Product Description

CVNC-T2 generates part programs for the following:

- 2-axis continuous path profiling
- Rough turning
- Boring
- Facing at any angle
- 2-axis positioning
- Twin-turret support
- Centerline hole processing

CVNC-T2 supports single- or twin-turret, 2-axis turning equipment having either horizontal or vertical spindle orientation. Twin-turret equipment can have independent slides, but only one turret is displayed and programmed at a time.

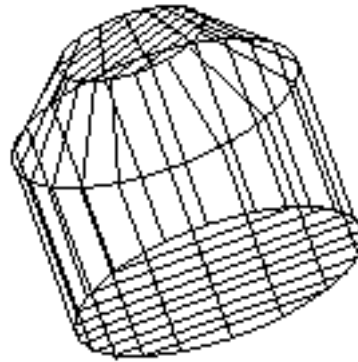
You can work in multiple construction planes (Cplanes) in a single field of orientation. The part programs generated are postprocessed to produce control tapes for 2-axis NC turning equipment.

CVNC-T2 references wireframe geometry part models constructed with any of the following CADDs entities:

- Points
- Lines
- Arcs
- Circles
- Ellipses
- Conics
- Parabolics
- Splines, B-splines, and Nsplines

CVNC-T2 generates tool motion conforming to the part model. See the following figure for an example of a part model.

Figure 1-1 Wireframe Part Model



Using CVNC-T2 commands, you control the sequence of events that defines, sets up, and executes a job containing a set of turning and/or hole processing operations.

Job Control File

A Job Control File (JCF) is the CVNC part program that records all inputs, including coordinate data and geometric entities. A JCF is an ASCII text file that can be saved, edited, and reexecuted. It may contain macros and pass-through statements.

What Is a JCF?

A JCF is a numerical control part program that stores a sequence of instructions to control the machine tool. It is a permanent NC part program, whose instructions include the following:

- CVNC commands
 - Macros
 - Pass-through statements
1. As you enter CVNC-T2 commands, the JCF is automatically generated, providing graphics as appropriate.
 2. As you execute each command, you can examine and verify the resulting tool motion.

Typically, a JCF corresponds to an operation or a single setup on a machine tool. The JCF consists of commands that result in output for machine control (such as CHGTOOL and MOVE) and tool motion (such as TURN TO OD, GROOVE, and THREAD).

A full set of editing commands is available to position within the JCF and make changes, such as changing, copying, and deleting lines. These edits are simultaneously reflected in the display of graphics. (See the *CVNC Editor Guide* for editor instructions.)

The beginning of a JCF is shown in the next figure.

JCF Example

```
CVNC TURN  HEADER LINE
#>
#> SET CPLANE AND TOOL CONFIGURATION AND POSITION
#>
DATUM 'TOP'
CONFIG REAR XDIF 4.0 YDIF -12.0
HOMEPT FROM  X 14 Y 9 ;
#>
#> SET USER DEFINED TOOL LIBRARY NAME
```

```
#>
TLIB 'T2.DEMO'
#>
#> DEFINE TOOL FOR ROUGH TURNING, PROFILING, GROOVING, THREADING,
#> DRILLING, AND BORING
#>
DEFTOOL 'TOOL.OD-ROUGH' TURN RAD .031 XGAGE 1.219 YGAGE 4.969
DEFTOOL 'TOOL.OD-PROFILE' TURN RAD .031 XGAGE 1.219 YGAGE 4.969
DEFTOOL 'TOOL.OD-GROOVE' TURN RAD .005 XGAGE 1.245 YGAGE 4.995
DEFTOOL 'TOOL.OD-THREAD' TURN RAD .005 XGAGE 1.188 YGAGE 4.995
DEFTOOL 'TOOL.ID-DRILL' DRILL DIA 1.5 ATANGL 120 XGAGE 4.0 YGAGE
0.0
DEFTOOL 'TOOL.ID-BORE' TURN RAD .031 XGAGE 4.0 YGAGE -.619
DEFTOOL 'TOOL.ID-GROOVE' TURN RAD .005 XGAGE 4.0 YGAGE -.688
DEFTOOL 'TOOL.ID-THREAD' TURN RAD .005 XGAGE 4.0 YGAGE -.688
#> DEFINE OUTPUT FILE FORMAT
#>
OUTPUT CLFILE TEXT 'T2.SAMPLE'
#>
#> SET PPRINT STATEMENT ON
```

How Are JCFs Created?

You can create a JCF in the three ways.

Interactive : Commands are immediately verified for correct syntax, executed, and stored in the JCF. Motion commands result in graphic display on your screen.

Syntax Verification: Commands are checked for correct syntax and stored in the JCF, but they are not executed immediately. JCF command execution can be performed at a later time, perhaps during off-hours.

Offline : Commands are written into a file using a text editor. They can subsequently be executed interactively or in batch mode. There is no automatic syntax checking performed when this method is used. There is syntax checking when these commands are executed.

CVNC-T2 Features

CVNC-T2 includes a default language environment that consists of CVNC commands, command files, and macros for turning and hole processing. You also receive a set of output pass-through statements. You can modify the language environment with job-specific macros or with new output pass-through statements. You can examine system variables containing machining and system parameters at any time.

System Variables

CVNC-T2 maintains a large number of system variables containing machining and system parameters that result from the execution of JCF commands. These parameters are accurate with respect to your current line position in the JCF. You can reference system variables to find current values, such as the following:

- Active feed rates
- Spindle speeds
- Radius or name of the active tool

CVNC displays current system variable values in the Status window of your workstation screen. For a system variables listing, see the *CVNC System Variables Guide* or enter #! at the NC: > prompt.

System Monitoring Tools

You can set up the CADD5 Status window to continually display and update the system variables you want to see. For more details see the SHOW, PRINT, ASSIGN, and STATUS commands described in the *CVNC System User Guide and Menu Reference*.

Macros

Macro commands allow you to do control, branching, and looping functions in your JCF. Several CVNC commands included in the CVNC-T2 application package are macros. You can also write your own macros in CVMAC.

With macros, you can customize and enhance your interface and programming environment. For more information, see *Customizing CVNC and CVMAC Language Reference*.

Command Files

Command files are text files containing a sequence of CVNC commands that perform repetitive functions such as moving to the home point or changing the tool. Command files contain no logic. You can write command files as standalone executable files and call them as needed during any NC process. For more information, see the *CVNC Editor Guide*.

Output

CVNC-T2 converts the data in the JCF to one of three output formats supported by CVNC:

- APT
- CLFile
- COMPACT II

Specify the output format needed with the OUTPUT command. APT and COMPACT II output must be processed offline. CLFile output can be processed online or offline.

The output generation process normally executes in the background separate from the CVNC-T2 work session. This allows you to be productive in CADDSS as soon as you leave a CVNC work session.

For more information about output generation, see the *CVNC System User Guide and Menu Reference*.

Output Pass-through Statements

Pass-through statements are postprocessor commands included in the CVNC grammar files. You can enter pass-through statements in the JCF in the same manner as CVNC commands.

Pass-through statements are not processed by CVNC. Instead, they pass through to APT, CLFile, or COMPACT II output. Available pass-through statements are listed in Appendix A of the *CVNC-T2 Command Reference*.

Included in the CVNC-T2 application package are several output pass-through statements modified from the APT, CLFile, and COMPACT II languages. Pass-through statements do not affect CVNC processing, but they are sometimes needed to control output file processing or the machine tool. They can be used like CVNC commands in your JCF. CVNC-T2 passes the values directly to the output file.

You can generate additional pass-through statements based on APT/CLFile or COMPACT II commands using CVNC features that append statements to the CVNC language.

Modifying the CVNC-T2 Language Environment

The CVNC-T2 language is defined by grammar files external to CADDs. You can reconfigure these files at the operating system level to integrate job-specific macros or new output pass-through statements into the language.

When you enter CVNC-T2, the active grammar is in these two files:

```
/usr/apl/cadds/data/cam/ncturn/grammar/_grm  
/usr/apl/cadds/data/cam/ncturn/grammar/_voc
```

These files contain base grammar, macros, and APT/CLFile pass-through statements.

An alternative pair of grammar files is available for COMPACT II output. Use the LANGUAGE command to activate these files:

```
/usr/apl/cadds/data/cam/ncturn/grammar/compact/_grm  
/usr/apl/cadds/data/cam/ncturn/grammar/compact/_voc
```

A third pair of grammar files contains base CVNC grammar only. Working in these files, you can alter, replace, or add to the macro definitions and pass-through statements provided with CVNC:

```
/usr/apl/cadds/data/cam/ncturn/grammar/base/_grm  
/usr/apl/cadds/data/cam/ncturn/grammar/base/_voc
```

Modifications to these files can be permanent or temporary. You use the `ncgram append` command at the operating system level to append the grammar file you want to modify. An interactive macro utility (called `macutil`) allows you to automatically append the CVNC-T2 grammar file you specify.

For more information about these procedures, see *Customizing CVNC*.

Using CVNC-T2

This chapter explains how to enter CVNC-T2 and build your JCF.

- Entering CVNC-T2
- Entering and Editing Data

Entering CVNC-T2

Enter CVNC-T2 using the CADD5 PROGRAM NCTURN command. Once you are in the turning module, CVNC displays the NC: > prompt.

Perform CVNC-T2 turning operations on parts designed with CADD5 entities. Tagged entities are shown in Figure 2-1.

Use the following modifiers with the CADD5 PROGRAM NCTURN command:

- EXEC - Reexecutes the contents of the entire existing JCF, regardless of the last filed position.
- TOP - Places you at the first line of the existing JCF.
- NOGRAPH—Turns off tool path graphics.
- filename - Specifies the name of a new or existing JCF.
- OPR exp - Specifies the JCF by an operation number.
- EXT text - Specifies a new JCF file name as an extension to an existing JCF file name.
- : ent d1 - Identifies a tool path for an existing JCF.
- PROTECT - Protects all CVNC referenced and created entities from deletion.

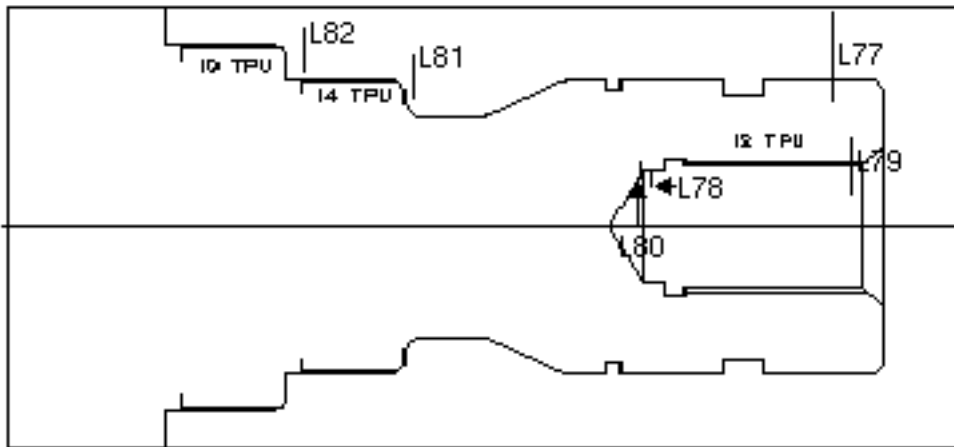
Develop your part program and the corresponding JCF by executing CVNC commands. Enter commands using the keyboard or the mouse.

The input sequence is generally command-modifier-value. Some modifiers require many values, others none. Still other modifiers require entity or location selections. CVNC checks syntax word by word and traps typographical errors at once.

CVNC-T2 processes the command immediately, updating appropriate system variables and graphics.

For more information about the PROGRAM command, see the *CVNC System User Guide and Menu Reference*.

Figure 2-1 Tagged Entities



Entering and Editing Data

Develop your part program and the corresponding JCF by executing CVNC commands. Enter commands by typing them at the keyboard or by using the CVNC interface of icons, menus, and property sheets.

Entering Coordinate and Entity Data

Indicate a location either by using the mouse to digitize the location on your graphics display or by specifying explicit coordinates. Indicate an entity by either digitizing it or specifying its entity tag. The first character of an entity tag is preceded with a dollar sign (\$).

To digitize coordinate data in CVNC, you must be in `getdata` mode. Enter `getdata` mode by typing a colon (:) in the command line. Exit `getdata` by typing a semicolon (;). The system automatically converts entity digitizes to entity tags and location digitizes to explicit coordinates.

Please note: You can enter the name of a predefined NCGROUP whenever `locs` or `ents` are expected.

For example, to establish a series of three entities as an NCGROUP called `GROUP1` and to indicate the entities with digitizes, enter the following command at the `NC:>` prompt:

```
NCGROUP GROUP1 ENT: d1 d2 d3 ;
```

CVNC responds as follows:

```
NCGROUP GROUP1 ENT $L1 $L2 $L3 ;
```

To use entity tags instead of digitizes to define `GROUP1`, you can enter the following command:

```
NC:> GROUP GROUP1 ENT $L1 $L2 $L3 ;  
NCGROUP GROUP1 ENT $L1 $L2 $L3 ;
```

Please note: Delimit input coordinate or entity data with a semicolon. Whenever you specify information with entity tags or explicit coordinates (with or without `getdata`), terminate the data with a semicolon. For more information on entering coordinate and entity data, see the *CVNC System User Guide and Menu Reference*.

Editing JCF Data

A backup facility in CVNC allows you to undo commands and reposition yourself at a previous line in the JCF.

When you use the backup facility, the graphics disappear and system variables are reset to earlier values as appropriate. You can then modify the JCF at the specified line, execute your corrections, and move forward again over the commands you backed over (which have been retained by CVNC).

The values and graphics associated with these commands adjust themselves relative to the changes you have made.

See the *CVNC Editor Guide* for more information.

This chapter explains how to set up the machine and tool configurations.

- Configuring the Machine
- Defining Tool Parameters

Configuring the Machine

To configure the machining environment:

1. Use the DATUM command to identify the Cplane that establishes the machine coordinate system and machine zero. All data points in the output file are given with respect to the coordinate system of the DATUM Cplane.
2. Use the HOMEPT FROM command to identify the starting location and the home location of the cutting tool. This position is typically used for the location of tool changes.
3. If necessary, use the CONFIG command to define the positions of and the orientation between the primary and secondary turrets.

Using DATUM

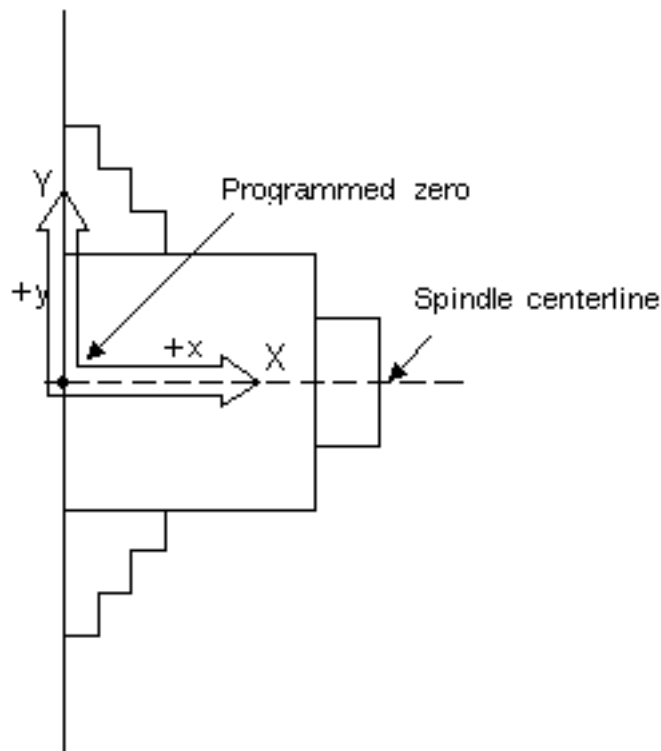
Use the DATUM command to identify the Cplane on the CADDs part model that defines the location and orientation of the machine coordinate system and machine zero, as shown in Figure 3-1.

The specified Cplane also becomes the currently active Cplane.

Do not use DATUM to identify your active Cplane. Use the CPL command to select construction planes during part programming. Also use CPL to activate a Cplane.

```
DATUM `TOP`
```

Figure 3-1 DATUM Command



Using HOMEPT FROM

Use the HOMEPT command with the FROM modifier to specify a HOME location for the turret. HOMEPT FROM sets the system variables #HOMEPT (home point) and #CURLOC (current tool location).

When defining HOMEPT FROM, consider that the #HOMEPT location should accommodate the greatest tool gage length to be used, as well as space requirements for loading and unloading the part, indexing the turrets, and time constraints for the job being performed.

Define HOMEPT FROM before executing the CHGTOOL command (see Chapter 4, “Operation Setup”). This starting home point specifies where the tool gage length reference point is currently located.

After you execute CHGTOOL, tool motion is displayed from the new tool tip location (#CURLOC). CVNC adjusts #HOMEPT and #CURLOC for the new tool x- and y-gage lengths.

The x-gage length and the y-gage length are measured along the x-axis or y-axis, respectively, in a positive or negative direction from the active tool radius center to the gage length reference point.

```
DATUM 'TOP'
HOMEPT FROM X 14 Y 9
DEFTOOL 'TOOL.OD-ROUGH' TURN RAD .031 XGAGE 1.25 YGAGE 5.0
CHGTOOL REAR CLW 1 'TOOL.OD-ROUGH' SFIG
```

Figure 3-2 HOMEPT

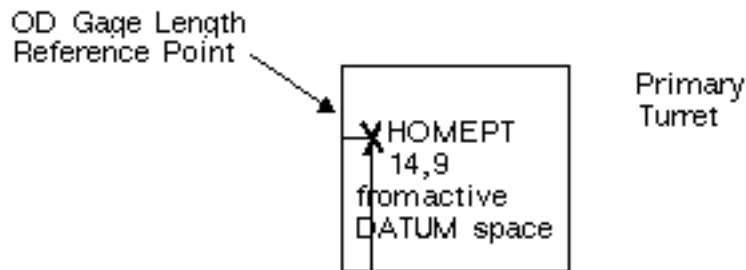
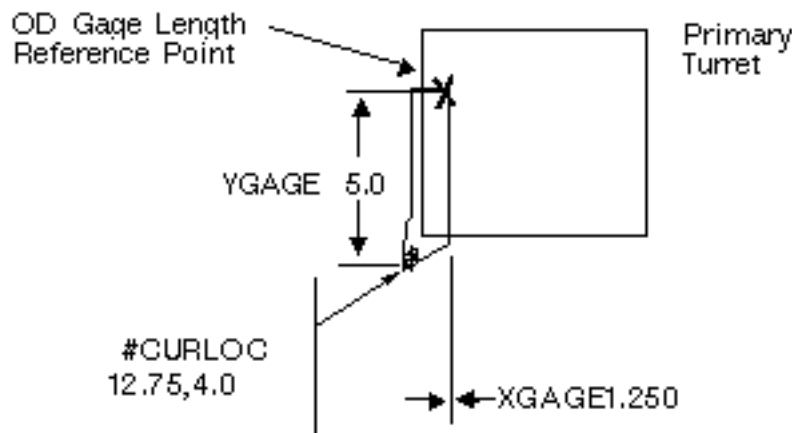


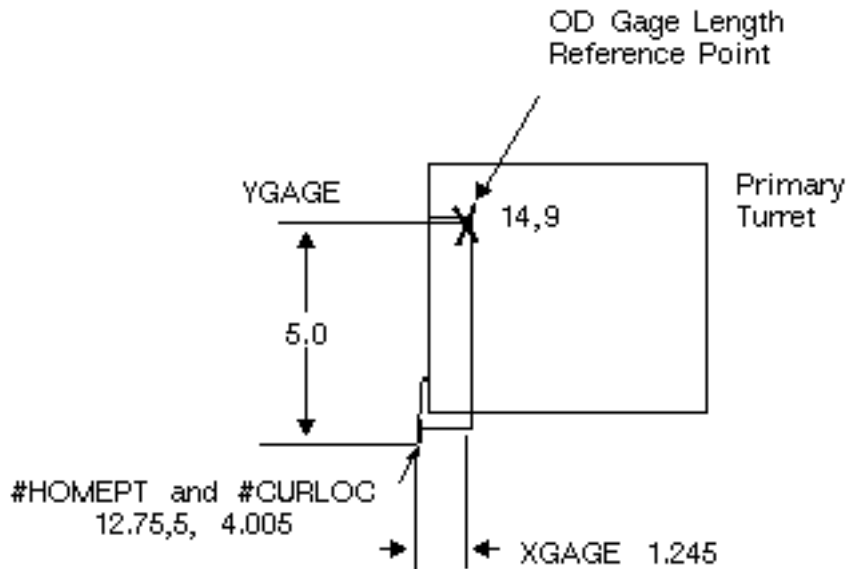
Figure 3-3 After First CHGTOOL



When a subsequent CHGTOOL is executed, tool motion is displayed from the new tool tip location. #HOMEPT and #CURLOC are again adjusted by the difference in XGAGE and YGAGE dimensions.

```
DEFTOOL 'TOOL.OD-GROOVE' TURN RAD .005 XGAGE 1.245 YGAGE 4.995
CHGTOOL REAR CLW 6 'TOOL.OD-GROOVE' SFIG
```


Figure 3-4 After Second CHGTOOL



Using CONFIG

Use the CONFIG command for twin-turret NC machines to define the distance from the primary to the secondary turret.

Figure 3-5 shows a twin-turret configuration with the primary turret holding outside-diameter (OD) tools and the secondary turret holding inside-diameter (ID) tools.

Often the primary and secondary turrets are fixed to a common slide that provides movement toward and away from the part.

Use the XDIF and YDIF modifiers to describe the distance and orientation between the two turrets. Distances are measured along the x- and y-axes, in a positive or negative direction, from a point on the primary turret to a point on the secondary turret. These points are called the gage length reference points, and are set by the machine tool user.

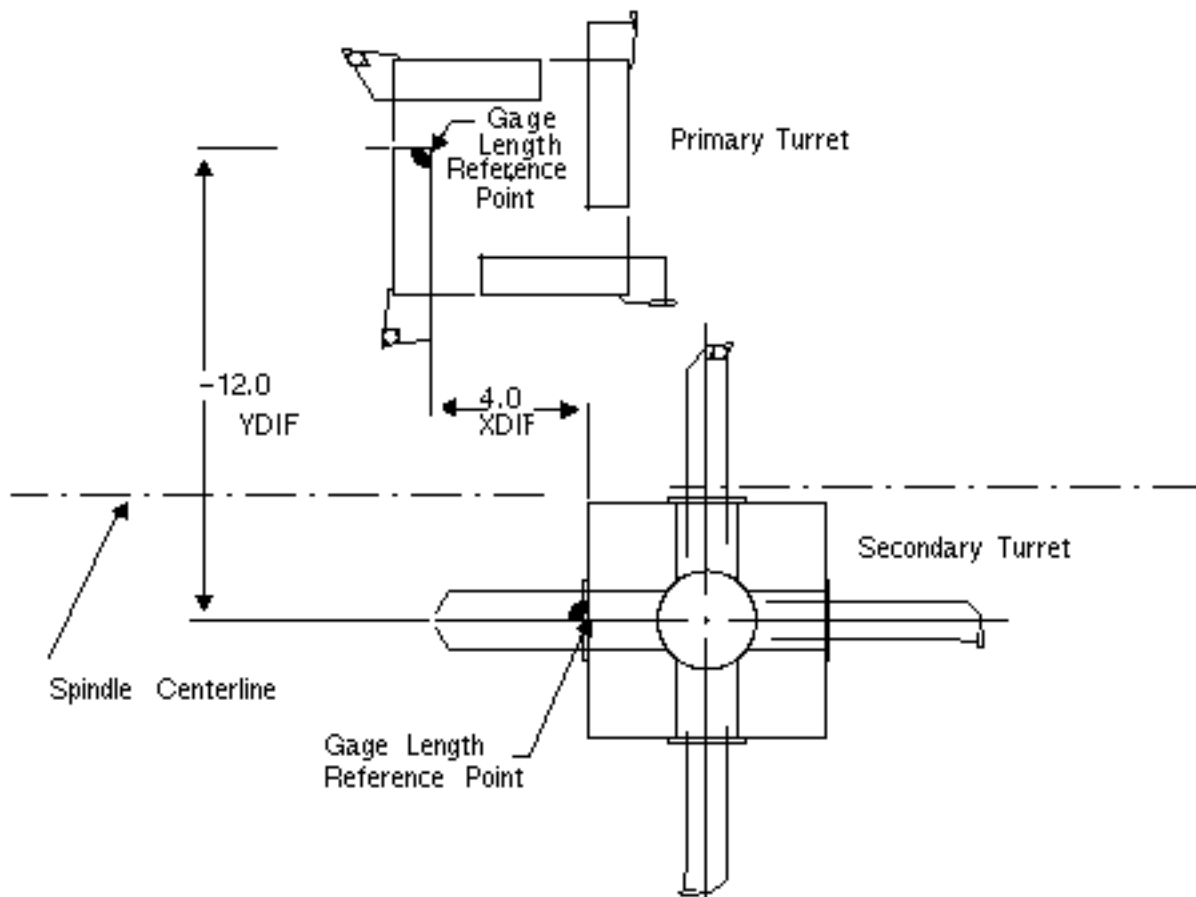
```
NC: > CONFIG REAR XDIF 4.0 YDIF -12.0
```

This figure also shows how the distance between active tools in a twin-turret configuration is defined.

Usually, as in this example, the gage length reference point on the OD turret is coincident with the corner of the tool holder diagonally opposite the center of the active tool.

The gage length reference point on the ID turret may be coincident with a point on the centerline of the tool holder. In this example, that point is coincident with the intersection of the tool holder centerline and the face of the ID turret.

Figure 3-5 Twin-Turret Configuration



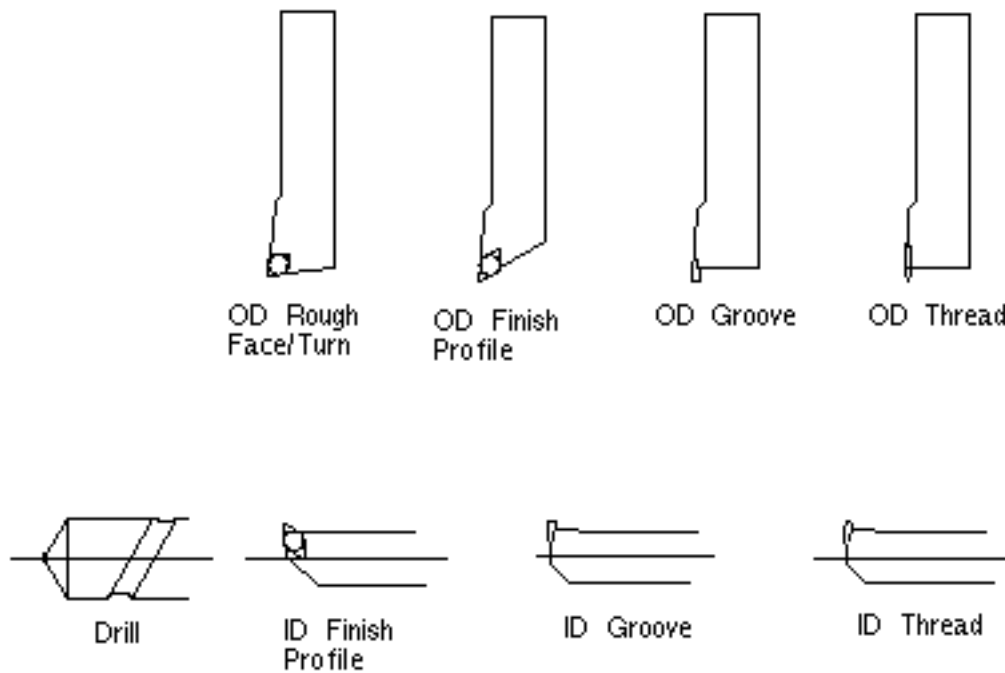
Defining Tool Parameters

Use the DEFTOOL command to specify the tools and tool parameters you need for a given operation. CVNC-T2 supports tools for turning, boring, facing, drilling, and tapping operations.

Tool parameters include tool radius (or diameter) and x- and y-gage lengths.

Turning operations include rough and finish turning, boring, facing, grooving, and threading. Here are examples of turning tools in OD and ID tool holders.

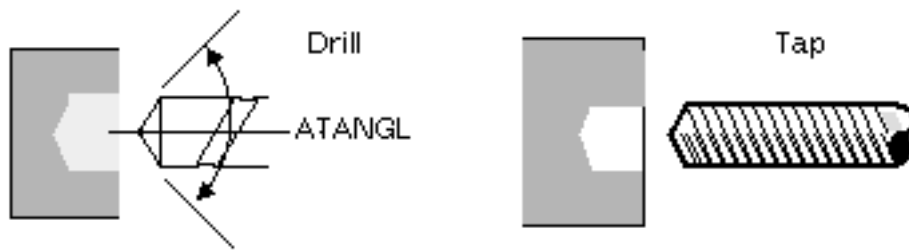
Figure 3-6 OD and ID Tool Examples



When you specify a drill, you must indicate the drill angle at the cutting end of the tool; use the ATANGL modifier with DEFTOOL to do this. When you specify a tap, indicate the number of threads per database unit; use the TPU modifier with DEFTOOL to do this.

Here are examples of a drill and a tap.

Figure 3-7 Drill and Tap Tools



Operation Setup

This chapter explains how to select the tool and set various operation parameters.

- Selecting a Tool and Changing the Active Cplane
- Specifying Feed Rate and Spindle Speed
- Setting STOCK Offsets
- Specifying Tolerance
- Changing the Tool Gage Length

Selecting a Tool and Changing the Active Cplane

Selecting a Tool

Use the CHGTOOL command to select a predefined cutter and the LISTOOL command to display tool parameters.

Use CHGTOOL to select a predefined tool from the active tool library for an individual turning or hole processing operation.

Use LISTOOL to display the parameters of either an individual tool or all the tools in your tool library.

Changing the Active Cplane

The CPL command allows you to change the active Cplane or generate a new Cplane for easier part programming while you are developing an NC tool path.

Use the CPL command to do the following:

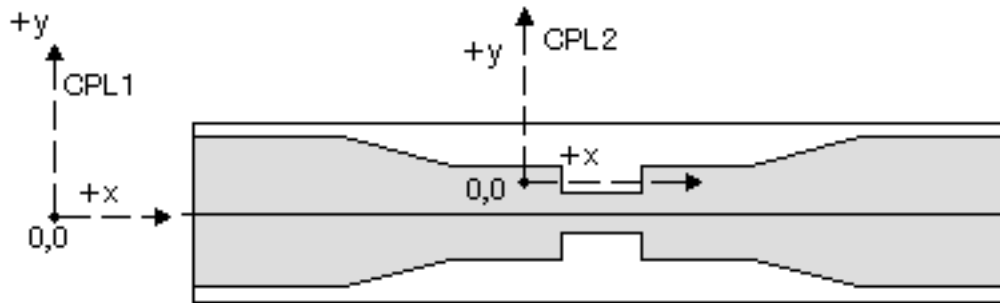
- Change the active Cplane to another existing Cplane
- Offset an existing Cplane to create a new Cplane
- Digitize the origin of a new Cplane

Changing the Active Cplane to Another Existing Cplane: Use the CPL command to change the active Cplane to another Cplane.

Change from the current Cplane, called CPL1, to another Cplane defined in CADD5 and called CPL2, by executing the CPL command, as shown in the following figure.

Figure 4-1 Changing the Active Cplane

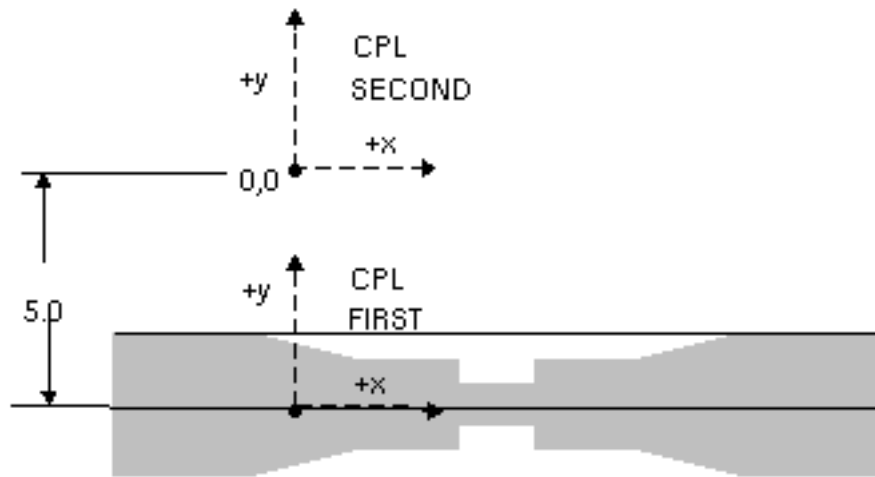
NC :> CPL "CPL2"



Offsetting an Existing Cplane: Use the CPL command to offset an existing Cplane to create a new Cplane, as shown in Figure 4-2.

Figure 4-2 Offsetting an Existing Cplane

NC :> CPL "FIRST" YCPL 5 CPL "SECOND"

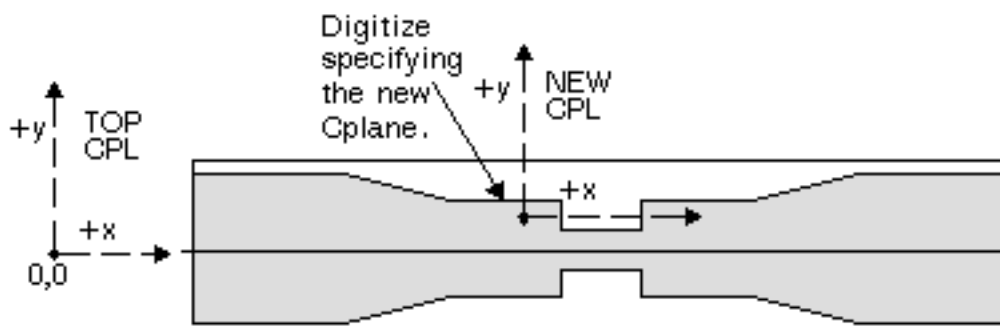


Digitizing the Origin of a New Cplane: Use the CPL command to digitize the origin of a new Cplane.

Digitize a location that indicates the origin of the new Cplane, as shown in Figure 4-3.

Figure 4-3 Digitizing the Origin of a New Cplane

NC:>CPL "TOP" MODEL Loc d CPL "NEW"



Specifying Feed Rate and Spindle Speed

Speed Rate

Use the SPEED command to specify the spindle speed for the following:

- Turning
- Threading
- Hole processing operations

You can select a modal speed range (HIGH, LOW, MEDIUM, or AUTO) particular to your NC lathe and indicate the desired spindle speed in the following:

- Revolutions per minute (RPM)
- Surface feet per minute (SFM)
- Surface meters per minute (SMM)

The default speed is zero.

Feed Rate

Use the FEED command to select the feed rate at which the tool moves during cutting or noncutting motion.

Modal Feed Rates

All motion is associated to modal feed rates. Modal feed rates are activated for a motion generated by a cutting command or a noncutting command. CVNC uses default modal feed rates if you do not define values for them with the FEED command.

Modal feed rates are activated for a motion generated by a cutting command (such as CUT or TURN) or noncutting command (such as MOVE) by the following motion types: APPROACH, PLUNGE, CUT, STEPIN, STEPOUT, POSITION, RETRACT, and CLEAR.

A single high-level motion command can generate several modal feed rates. For example, the TURN command will generate APPROACH, PLUNGE, CUT, STEPIN, POSITION and STEPOUT tool motion that will activate these corresponding feed rates.

CVNC uses the following default modal feed rates if you do not define values for them with the FEED command:

- APPROACH feed rate defaults to RAPID.
- PLUNGE feed rate defaults to the CUT feed rate.
- CUT feed rate defaults to zero.
- STEPIN and STEPOUT feed rates default to zero.
- POSITION feed rate defaults to RAPID.
- If you have set a value for the RETRACT feed rate, but not for the CLEAR feed rate, the CLEAR feed rate defaults to the RETRACT feed rate value.
- If you do not set a value for either the RETRACT or CLEAR feed rate, they default to RAPID.
- If you have set a value for CLEAR feed rate, but not for RETRACT feed rate, RETRACT feed rate defaults to the CLEAR feed rate value.

These modal feed rates continue to default until you set them. The default feed rate behavior may or may not be suitable for your machining operations.

Setting STOCK Offsets

You can affect tool paths by setting desired offsets with the STOCK command.

Use the STOCK command to add thickness to an entity or boundary. You can specify different stock for different entities that are being machined by the same command.

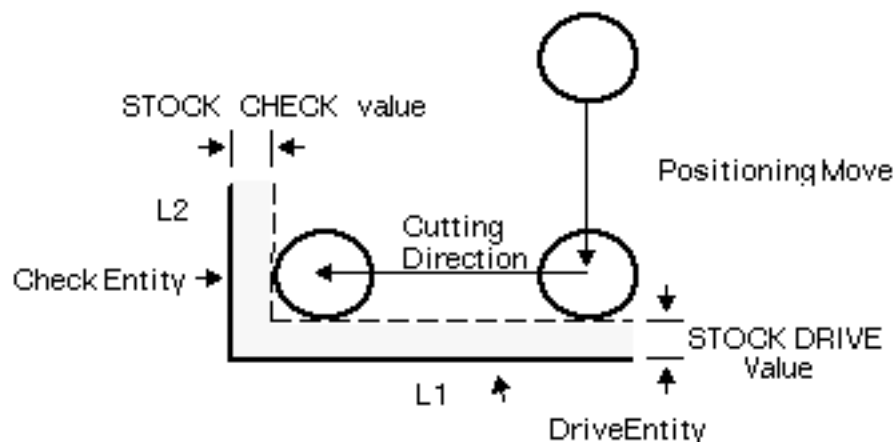
You can add stock values to a drive entity and a check entity.

- A drive entity is an entity along which the tool moves in a cutting operation.
- A check entity is an entity that a tool moves TO, ON, or PAST in a cutting operation.

The following figure shows how the STOCK command can be applied to each entity.

Figure 4-4 STOCK Values

```
NC:> STOCK DRIVE .30
NC:> STOCK CHECK .25
NC:> CUT TO $L1;
NC:> CUT CHECK $L2;
```



Add stock offsets to drive entities by executing the STOCK DRIVE command. STOCK DRIVE is used in the following operations:

- PROFILE
- TURN
- CUT (TO/ON/PAST)
- CUT [ENTITY]

Add stock offsets to check entities by executing the STOCK CHECK command. CUT [ENTITY] uses the STOCK DRIVE value for the entity being machined along the STOCK CHECK value to cut to or past the next entity.

After each CUT [ENTITY] command, the STOCK CHECK value becomes the STOCK DRIVE value. You can change STOCK CHECK between CUT [ENTITY] commands.

If another CUT ENTITY CHECK is issued, it becomes the stock value for the check boundary. This value is applied by CUT ENTITY to the selected check entity.

If another CUT ENTITY CHECK is issued, the check curve becomes the drive curve and this check stock becomes the interim drive stock (SHOW #IDRVTHK) for this operation only.

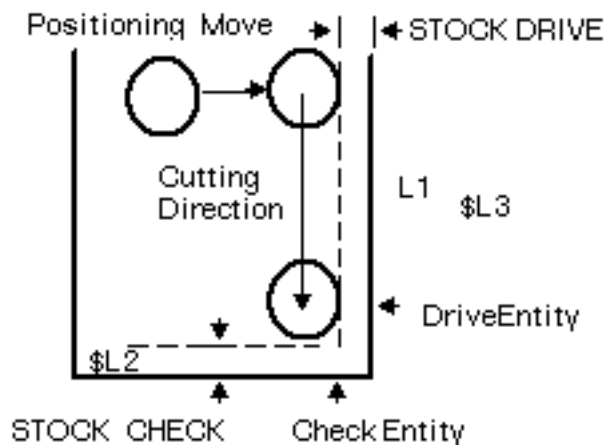
The drive stock value (SHOW #DRVTHK) is not changed.

Please note: If more than one stock value is specified for the same entity, the last specified value is used.

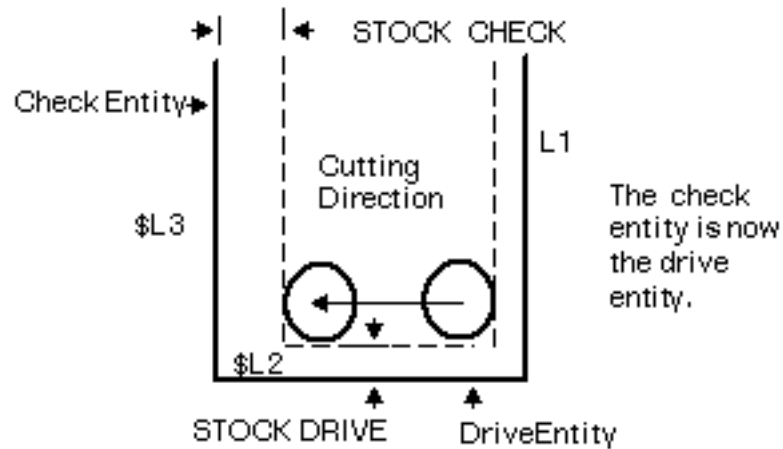
Positive stock is applied to the side of the cut and negative stock is applied to the opposite side.

Figure 4-5 Replacing STOCK CHECK with STOCK DRIVE

```
NC:> STOCK DRIVE .25  
NC:> STOCK CHECK .25  
NC:> CUT TO $L1  
NC:> CUT ENTITY CHECK TO $L2;
```



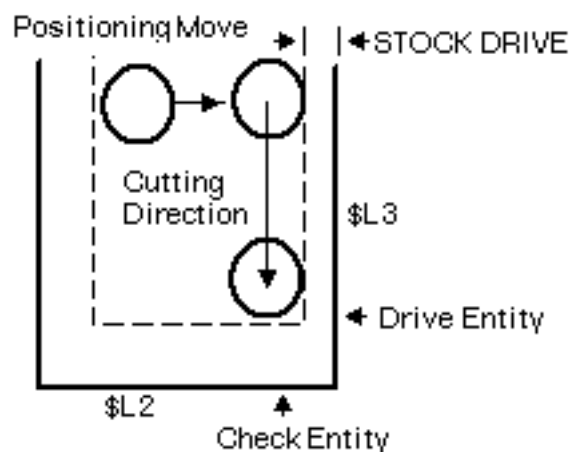
```
NC:> STOCK CHECK .50
NC:> CUT CHECK TO $L3;
```



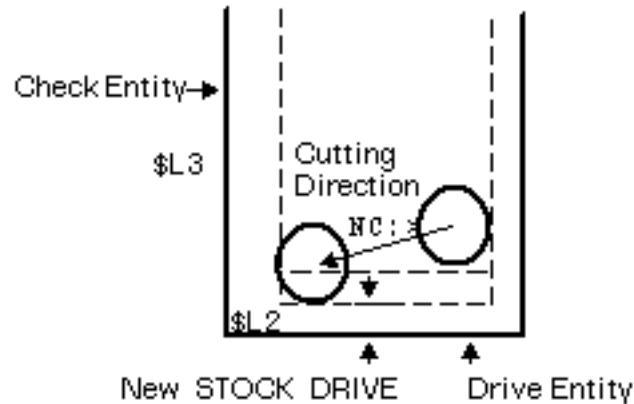
You can change STOCK DRIVE and STOCK CHECK between CUT [ENTITY] commands. Setting STOCK DRIVE between CUT [ENTITY] commands results in an angular move, as shown in the following figure.

Figure 4-6 Setting STOCK DRIVE Between Cuts

```
NC:> STOCK CHECK .25
NC:> CUT TO $L1;
NC:> STOCK CHECK .50
NC:> CUT ENTITY CHECK TO $L2;
```



```
NC: > STOCK DRIVE .25  
NC: > CUT ENTITY CHECK TO $L3;
```

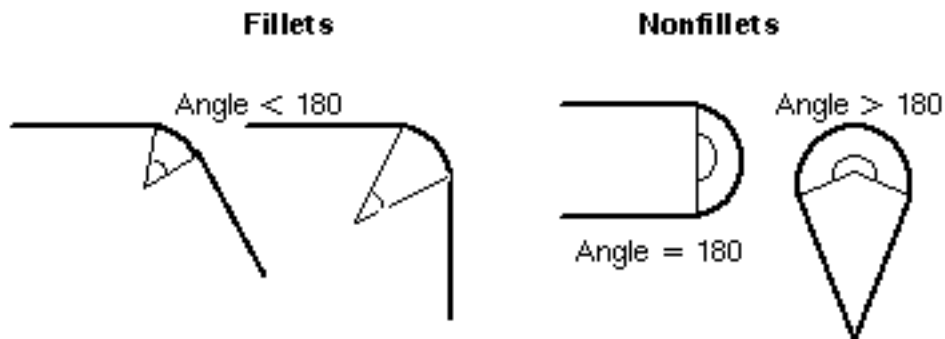


STOCK and Fillets

The STOCK command can also be used to specify stock for individual entities. STOCK can specify how fillets are treated when the entities they are adjacent to have stock assigned. In Figure 4-7, a fillet is defined as an arc less than 180° and tangent on both ends.

A fillet is defined as an arc less than 180° and tangent on both ends.

Figure 4-7 Fillets and Non-Fillets



Modifiers Used with STOCK

Use the ARCFLOAT and ARCFIX modifiers with STOCK to specify how machining behaves along boundaries that have fillets.

- ARCFLOAT floats fillets that do not have a stock value. This maintains tangency between the fillet and the rest of the boundary.
- ARCFIX specifies that the fillet is not to be floated.

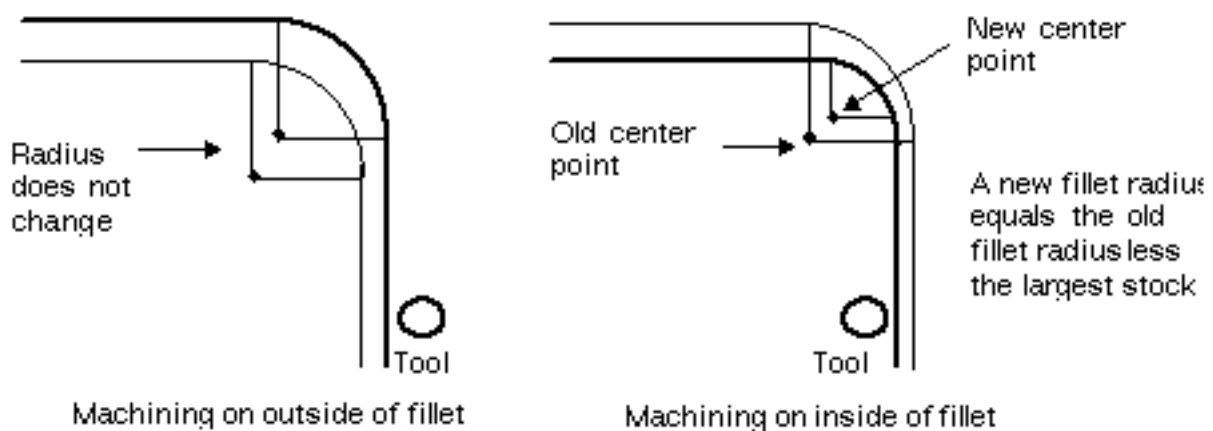
Fillet: Floated or Not Floated

Conditions when a Fillet is Floated

A fillet is floated as follows:

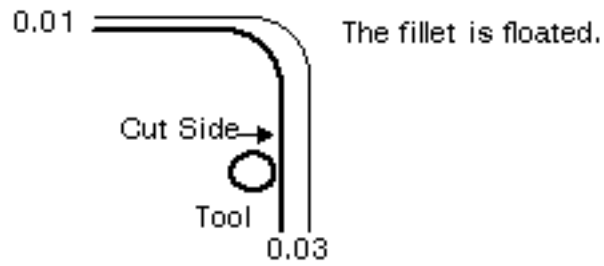
- When ARCFLOAT is selected and the fillet does not have stock.
- If machining is done on the outside of a fillet, the arc is floated in such a way that the radius remains the same and the center point changes.
- If machining is done on the inside of a fillet, the radius is reduced by the larger of the two stock values applied to the adjacent entities, and a new center point is determined.

Figure 4-8 Floated Fillet



Stock for entities adjacent to a fillet are applied to the same side as the cut.

Figure 4-9 Stock for Entities Adjacent to Fillet

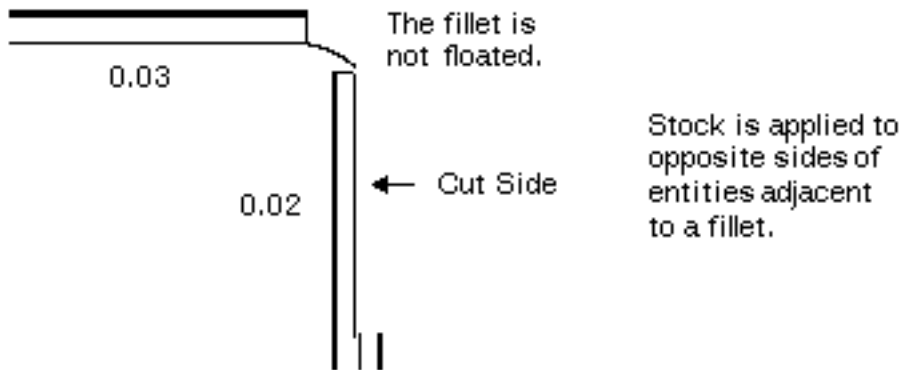


Conditions when a Fillet Is Not Floated

A fillet is not floated when

- ARCFLOAT is selected and the fillet has stock. The fillet is offset by the specified stock value.
- ARCFIX is selected.

Stock for entities adjacent to the fillet is not applied to the same side.



Specifying Tolerance

Use TOLER to set the inner and outer tolerance values for machining motion on a boundary or surface.

Use the INTOL modifier to specify the interior tolerance of the tool path. The interior tolerance is the amount the tool is allowed to cut into (beyond) the surface or boundary material at any point.

Use the OUTTOL modifier to specify the exterior tolerance. This is the amount of material allowed outside the surface or boundary, measured normal to the surface or boundary.

The tool path will stay within the INTOL and OUTTOL tolerance values.

For example, to specify an interior tolerance zone of .005 and an exterior tolerance of .0025, enter the following command:

```
NC:> TOLER INTOL .005 OUTTOL .0025
```

Please note: CVNC displays circular moves as a series of point-to-point moves, which stay within the INTOL and OUTTOL zones. This motion is written as circular interpolation records and as point-to-point data within a CLFile. Circular moves could be derived from circles, arcs, sections of curves, or from the CUT ARC command.

Postprocessors for machine tools that support circular interpolation do not usually use this coordinate data for driving the circular motion of the machine.

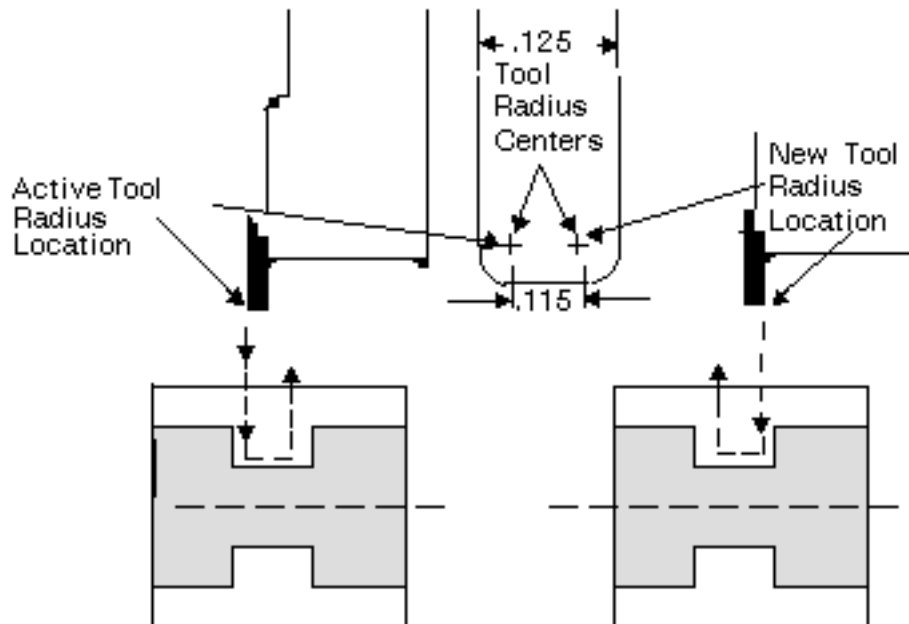
Changing the Tool Gage Length

Use the CHGAGE command to change the gage length of a tool when a different cutting edge of a tool is to be used.

Change the tool gage length by the distance between the tool radius centers of the cutting tool with CHGAGE. (You can use either the ABSL or INCR modifier.)

As shown in Figure 4-10, CHGAGE XGAGE INCR 0.115 alters the x-gage length by 0.115 so that the new tool radius location associated with the opposite cutting edge of the tool now becomes the active tool radius location. This feature can be used to finish machining a groove.

Figure 4-10 Changing Tool Gages to Cut a Groove



```
NC:> CUT INTOF  
NC:> CUT XINC  
NC:> MOVE  
NC:> CHGAGE XGAGE INCR -.115  
NC:> CUT INTOF  
NC:> CUT XINC  
NC:> MOVE
```

This chapter explains how to generate turning motion.

- Motion Types
- Rough Turning
- Profiling
- Threading
- Hole Processing
- Additional Motion Commands

Motion Types

Motion commands include cutting and noncutting movement. The noncutting motion commands for CVNC-T2 are the following:

- APPROACH - Motion toward the part to a position where a cutting operation begins
- RETRACT - Motion that disengages the tool from the part
- MOVE - Motion that repositions the tool

CVNC-T2 cutting motions can be divided into several categories, including

- Rough turning
- Profiling
- Threading
- Hole processing
- Individual motion commands used for both roughing and finishing

These operations are discussed next.

Rough Turning

Rough turning removes most of the excess material bounded by the material boundary and the part boundary.

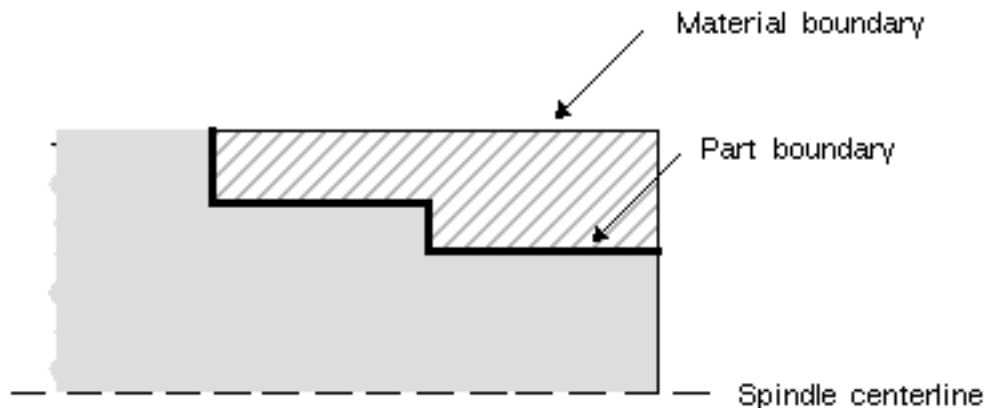
Use the TURN command for rough turning.

You can also use the PLUNGE and CUT commands, the GROOVE macro, as well as the FTURN macro for rough and finish turning for specific rough turning operations.

Note that part geometry for rough turning operations with TURN consists of two separate, continuous boundaries that together form a closed figure.

In Figure 5-1, the entities that outline the excess material around the part that must be cut bound the outside of the striped area and comprise the material boundary.

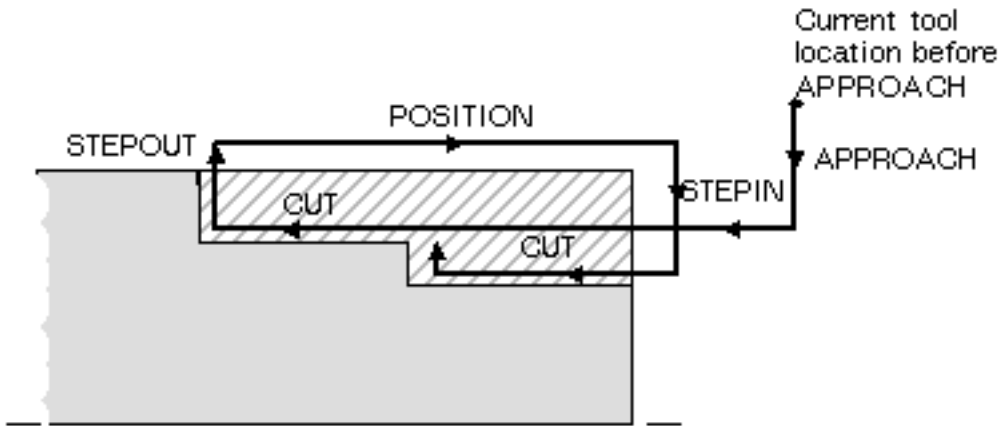
Figure 5-1 Part and Material Boundaries



The entities describing the geometry of the part after all the finishing operations have been completed comprise the part boundary.

The material and part boundaries are each separate and must be treated as such. Rough turning motions are shown in Figure 5-2.

Figure 5-2 Rough Turning Motions



Contoured and Noncontoured Approaches and Retractions

The default for the `TURN` command is contoured angles of approach and retraction. To change these defaults, use the `NONAPR` and/or `NONRET` modifiers, respectively.

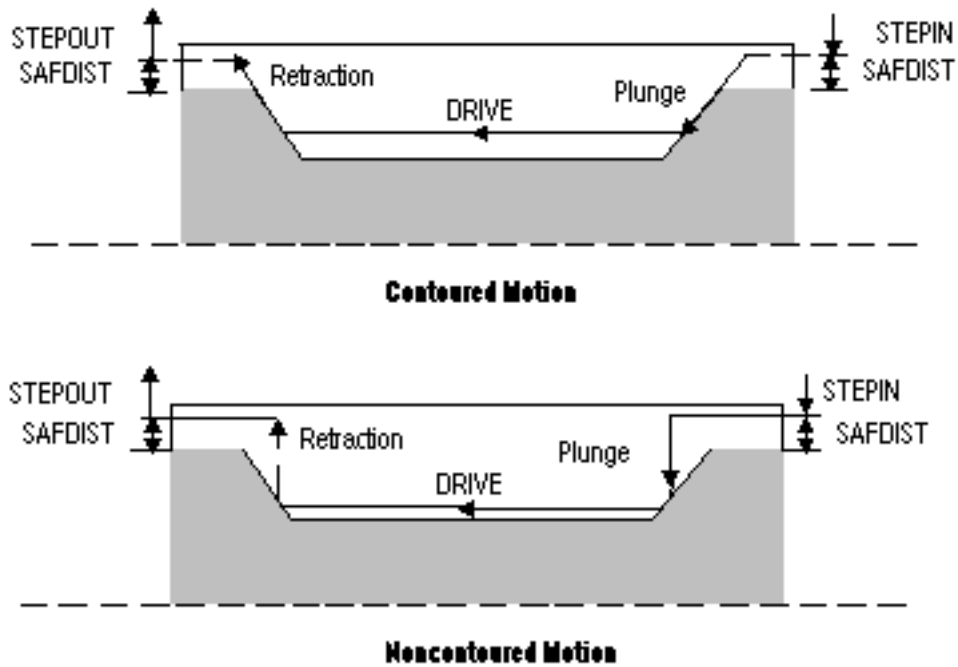
Contoured approaches and retractions move at the `CUT` feed rate and follow the contours of the boundaries at the beginning and end of the `DRIVE` vector.

Noncontoured approaches and retractions plunge and retract at the `STEPIN` and `STEPOUT` vectors to and from the cutting pass.

Use a reasonable `STEPIN` feed rate to prevent the tool from plunging into the material too quickly.

Distinctions between contoured and noncontoured motion are shown in Figure 5-3.

Figure 5-3 Contoured and Noncontoured Motions

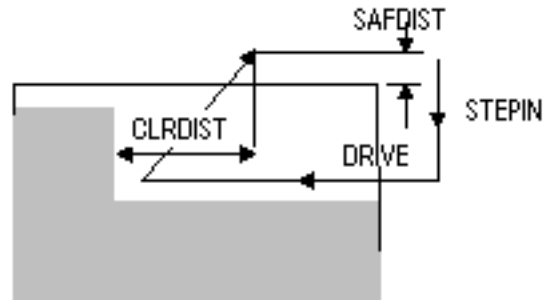


Specify an alternate STEPOUT distance for noncontoured retractions with the CLRDIST modifier. CLRDIST sets a clearance distance from the face (check boundary perpendicular to the drive).

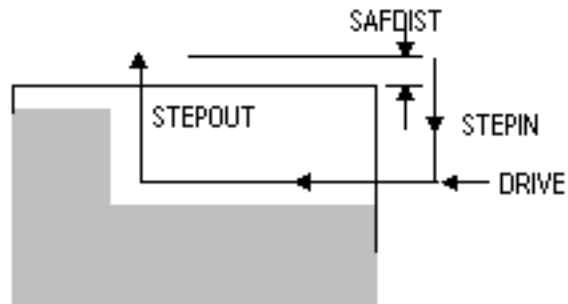
STEPOUTt with and without CLRDIST is shown in Figure 5-6.

In either case, the value is defined by the SAFDIST value from the material boundary of the previous cutting path.

Figure 5-4 Specifying CLRDIST



With CLRDIST



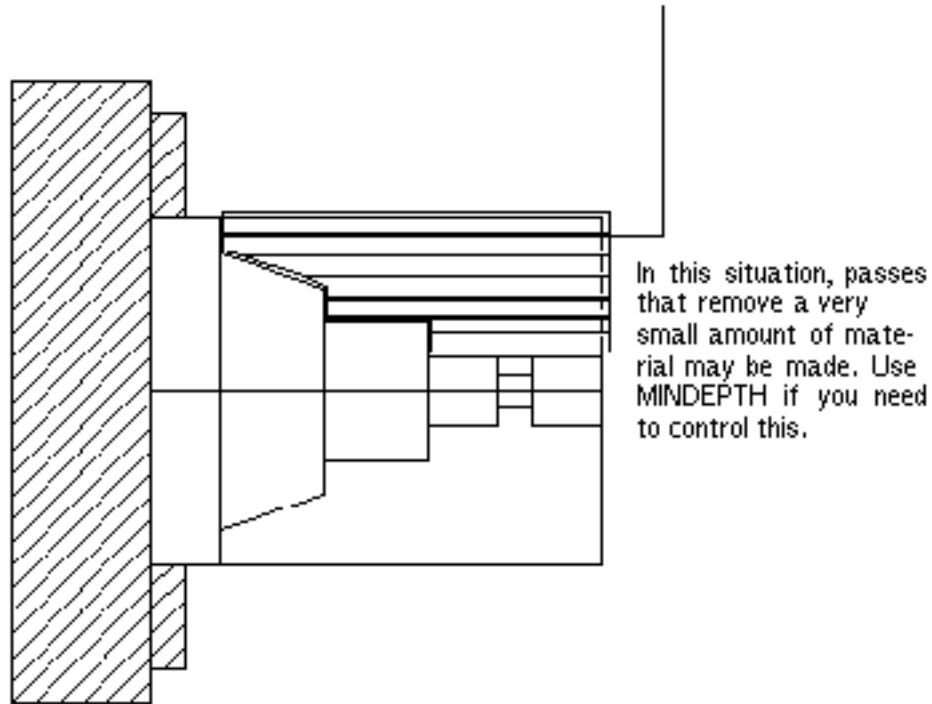
Without CLRDIST

Specifying Maximum Depth for Turning Motion

To optimize rough turning pass depth, specify a maximum depth (DEPCUT) value.

You can specify the maximum depth of each rough turning pass. Here is a turning operation with DEPCUT specified.

Figure 5-5 Turning with Only DEPCUT Specified



```
NC : > TURN $L21 $L54 $L18 $L40 $L32 $L12 $L12 $L35;MATL $L28 $L27: DEPCUT .3
```

Specifying Minimum Depth for Turning Motion

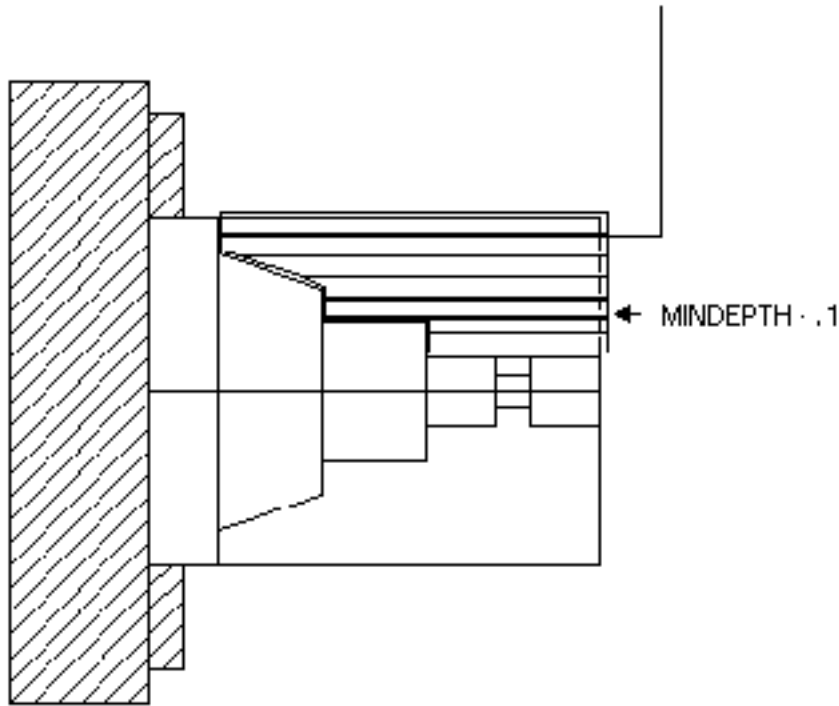
In addition to using DEPCUT, you can also specify the minimum depth of each rough turning pass with the MINDEPTH modifier.

Use the MINDEPTH modifier in situations where you need to control the minimum amount of material removed in each rough turning pass. Based on the input geometry, TURN attempts to ensure that the tool does not cut less than the specified MINDEPTH value, while not exceeding DEPCUT.

For some part geometries, it may not be possible for CVNC to generate passes with MINDEPTH between each pass. In these situations, CVNC issues a warning message. DEPCUT is not violated.

Figure 5-7 shows a turning operation when a MINDEPTH of .1 is specified. Note that in this example all the passes are within DEPCUT and MINDEPTH.

Figure 5-6 Turning with MINDEPTH and DEPCUT Specified



```
NC : >TURN $L21 $L54 $L18 $L40 $L15 $L32 $L12 $L35 ;MATL $L28 $L27 ; DEPCUT .3  
MINDEPTH .1
```

Undercutting During Rough Turning

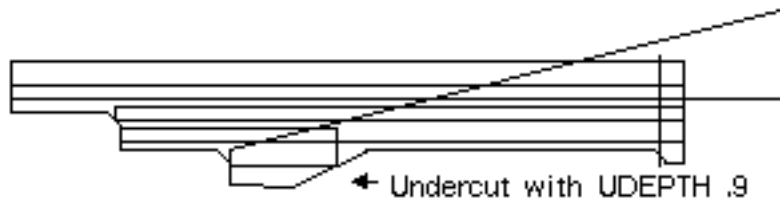
Each TURN command supports the machining of only one undercut.

TURN always cuts passes in descending order (toward the spindle centerline). Because of this, you can machine one undercut area only with a single command.

When the tool completes the undercut, it retracts to SAFDIST above the part. Follow this operation with another command, such as MOVE HOME, if more undercuts are required.

Figure 5-7 shows one undercut operation. Only the tool path for the TURN command including the undercut is shown.

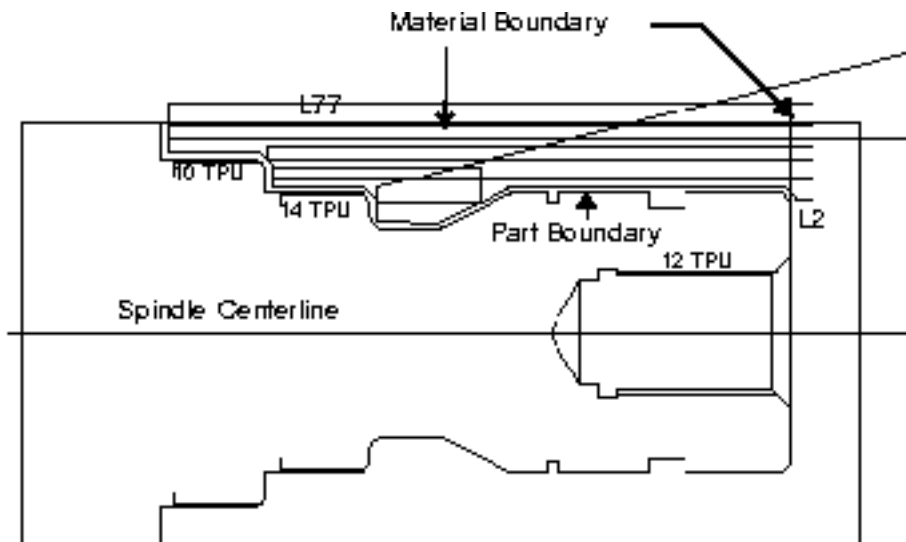
Figure 5-7 Tool Path for TURN Command Including Undercut



```
NC:> NCGROUP OD ENT $L22 $L68 $A3 $L10 $A2 $L9 $A1 $L8 $L20 $L7 $L6 $L21
$L5 $L12 ;
NC:> NCGROUP STK ENT $L77 $L2
NC:> TURN TO OD;MATL STK; DEPCUT .3 SAFDIST .2 UDEPTH .9
```

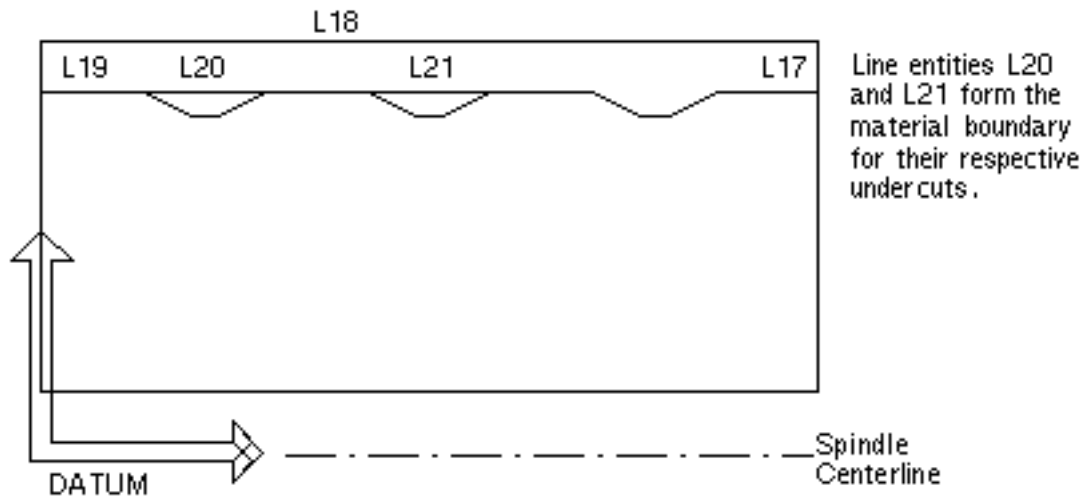
In Figure 5-8, the tool path is shown in relation to the material and part boundaries.

Figure 5-8 Tool Path in relation to Material and Part Boundaries



Here is a part with multiple undercuts. Only the first can be cleared in a single TURN. Enter a subsequent TURN for each remaining undercut. Also note that the two line entities (L20 and L21) form the material boundary for their respective undercuts.

Figure 5-9 Example of Three Undercuts in Sequence



Multiple Undercut Operation

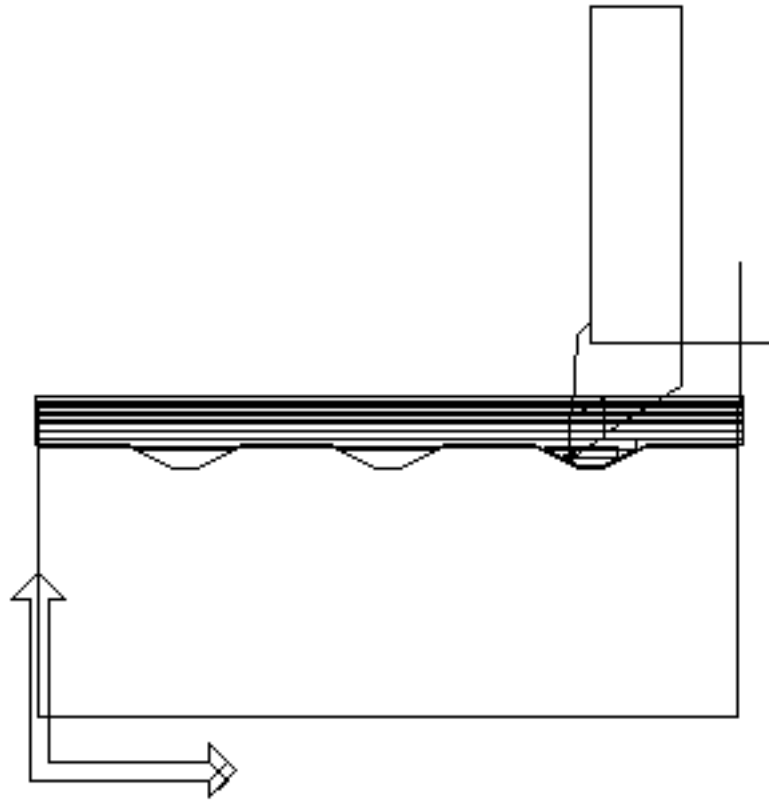
Figures 5-10 to 5-12 illustrate one possible solution to a multiple undercut operation using three separate TURN commands.

Job setup commands are the following:

```
NC:> DATUM 'TOP'  
NC:> HOMEPT FROM X 9 Y 10;  
NC:> CONFIG REAR XDIF 4.0 YDIF -12.0  
NC:> TLIB 'T2.DEMO'  
NC:> DEFTOOL 'TOOL.OD-PROFILE' TURN RAD .031 XGAGE 1.219 YGAGE 4.969  
NC:> CHGTOOL REAR 2 'TOOL.OD-PROFILE' SFIG
```

The first TURN clears away the bulk of the stock between the material and part boundaries and then turns the first undercut.

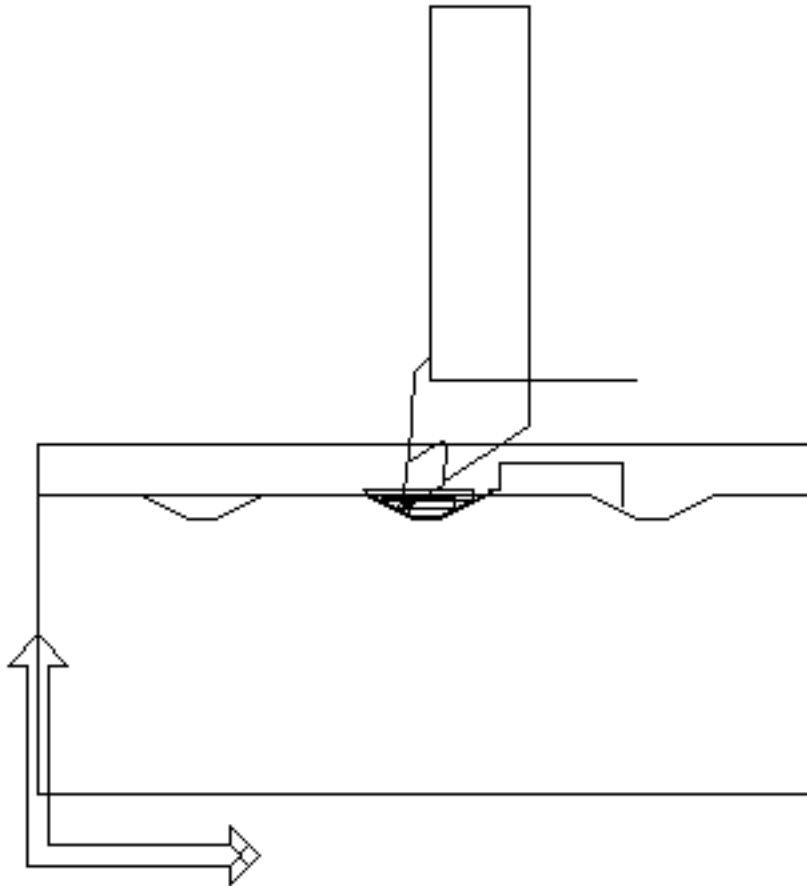
Figure 5-10 First Undercut of Three Undercuts in Sequence



```
NC:> SELECT LAY 2  
NC:> TURN TO $L14 $L13 $L12 $L11 $L10 $L9 $L8 $L7 $L6 $L5 $L4 $L3 $L2 ; MATL  
$L18 $L19 $L17 ;DEPCUT .1 UDEPTH .3
```

MOVE commands relocate the tool in the x- and y-coordinates before the TURN command for the second undercut.

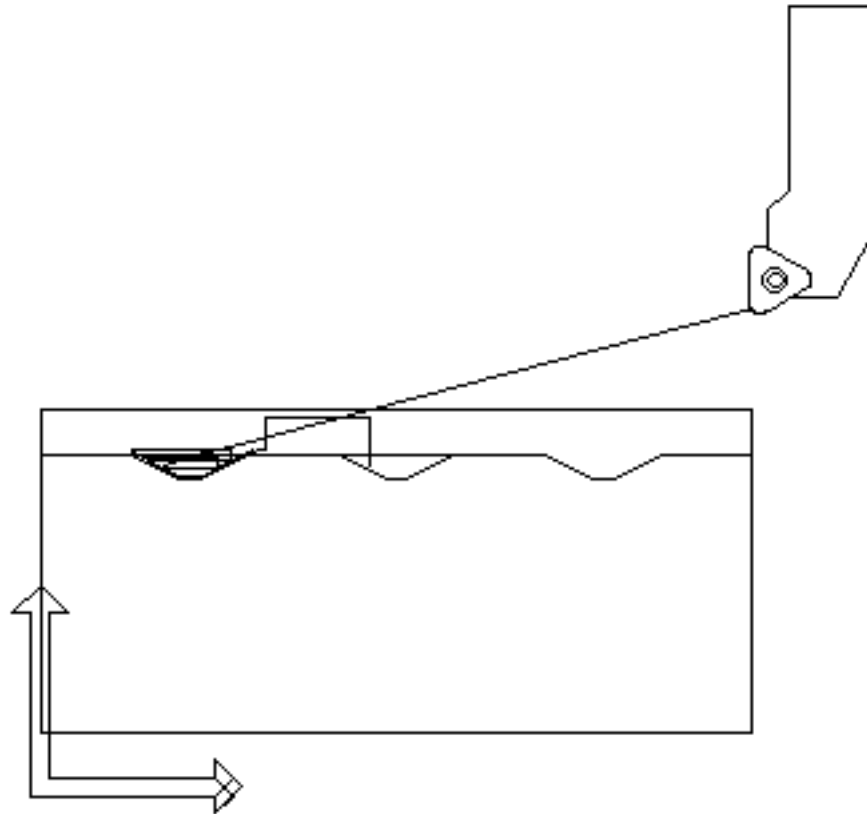
Figure 5-11 MOVE and TURN Commands for the Second Undercut



```
NC:> SELECT LAY 3  
NC:> MOVE YLOC X 5.79304 Y 3.31456 ;  
NC:> MOVE XLOC X 4.59225 Y 3.36998 ;  
NC:> TURN TO $L9 $L8 $L7 ;MATL $L21 ;DEPCUT .1
```

Again, the MOVE and TURN commands produce the third undercut at the desired location. MOVE HOME completes the operation.

Figure 5-12 The Third Undercut and MOVE HOME Command



```

NC:> SELECT LAY 4
NC:> MOVE YLOC X 3.72400 Y 3.42540 ;
NC:> MOVE XLOC X 2.44932 Y 3.40693 ;
NC:> TURN TO $L5 $L4 $L3 ;MATL $L20 ;DEPCUT .1
NC:> MOVE HOME
NC:> EXIT

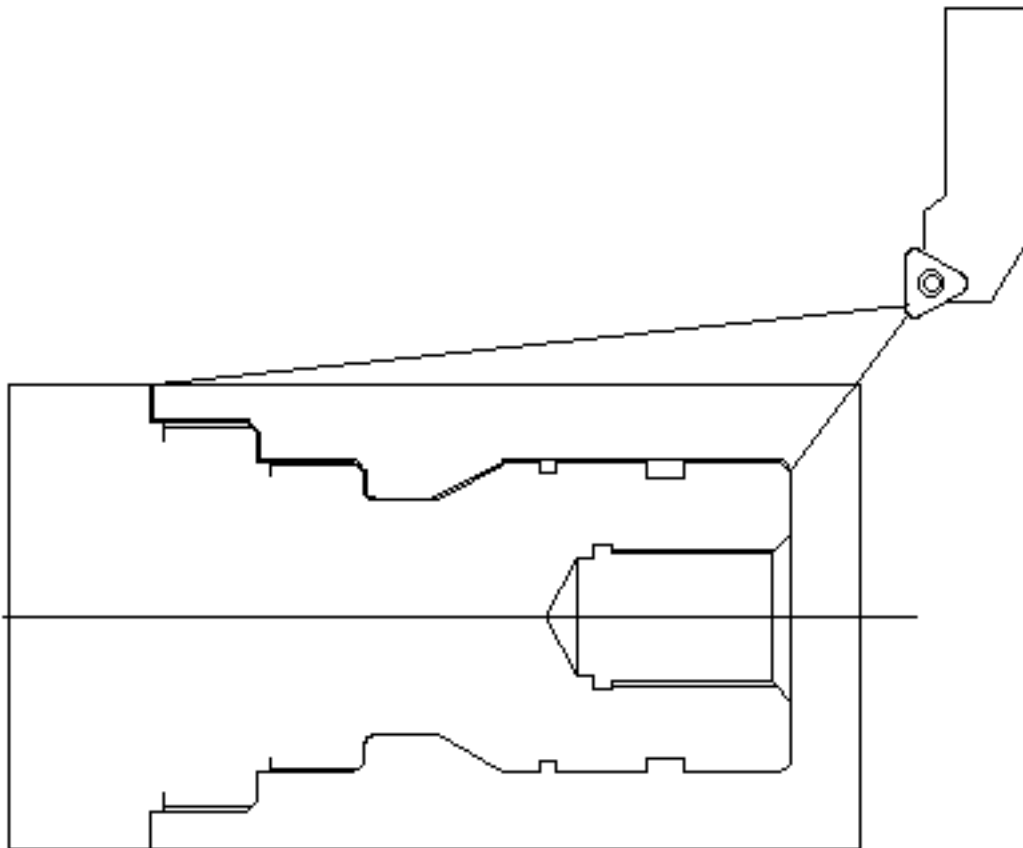
```

Note that the UDEPTH modifier was not used because the material and part boundaries define the TURN command tool cut for the second and third undercuts only. UDEPTH must, however, be specified for the first undercut.

Profiling

Finished turning is typically accomplished with the PROFILE command. PROFILE machines around a contiguous string of up to 512 part entities, treating the entire string as a single boundary. By treating the string of entities as a single boundary, PROFILE avoids narrow slots or corners where the tool does not fit. PROFILE does not allow changes in cutter compensation, or ON/TO/LEFT/RIGHT modes (as does CUT [ENTITY]). PROFILE does allow for varying stock values on the entities comprising the boundary. You can specify linear or radial extensions with the LEADIN and LEADOUT modifiers.

Figure 5-13 Profiling Example



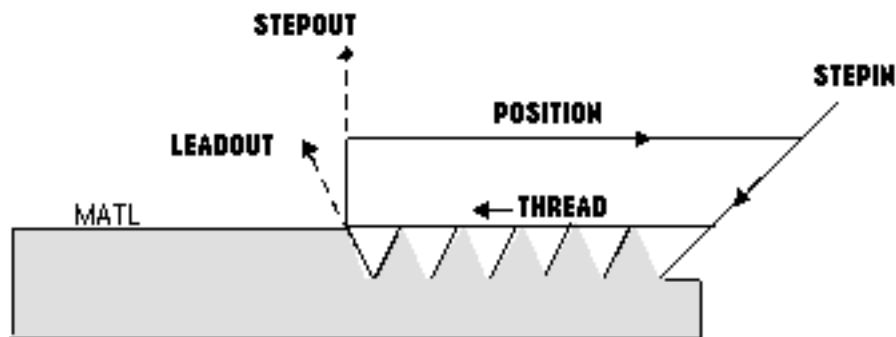
```
NC:> HOMEPT FROM X1 Y14 ;
NC:> NCGROUP OD ENT $L22 $L68 $A3 $L10 $A2 $L9 $A1 $L8 $L20
$L7
NC:> NCGROUP OD ENT $L22 $L68 $A3 $L10 $A2 $L9 $A1 $L8 $L20
$L7 $L6 $L21 $L5 $L12 ;
NC:> PROFILE TO OD; XYSIDE X 11.86409 Y 3.85379
NC:> MOVE HOME
```


Threading

Perform threading operations with the **THREAD** command. Use the **THREAD** command with the following operational setup commands for defining portions of threading motions:

- **STEPIN** - Specifies the vector along which the tool positions to the depth of each threading pass.
- **LEADOUT** - Specifies the vector along which each threading pass is terminated.
- **STEPOUT** - Specifies the vector along which the tool moves away from the part.

Figure 5-14 Threading Vectors



THREAD command modifiers define the following:

- Thread type
- Number of thread starts
- Thread location
- Threading passes
- Threading into and out of material
- Gaging passes

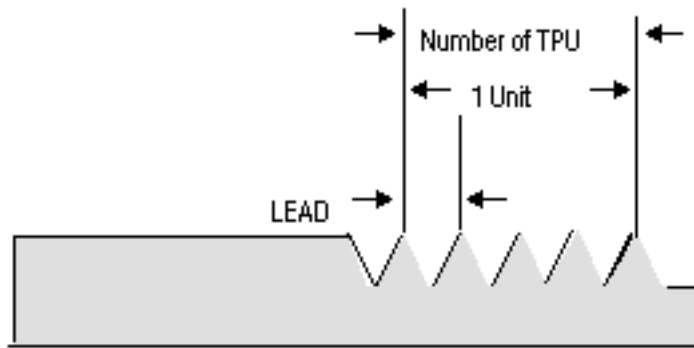
Thread Type

Use either the **TPU** or **LEAD** modifier to specify thread type.

Figure 5-15 shows a single start thread configuration that illustrates the relationship between the TPU and LEAD modifiers.

LEAD is defined as the distance between any point on a screw thread and the corresponding point on the next thread. In the case of multiple threads (NSTRT>1), each thread has the same value for LEAD.

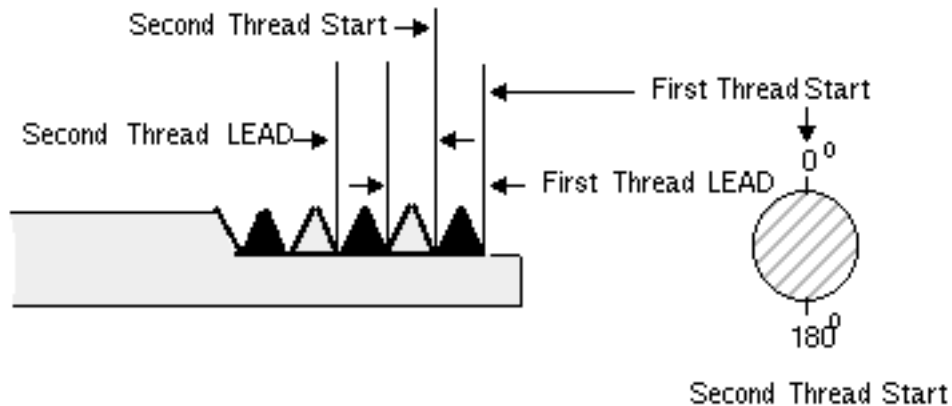
Figure 5-15 TPU and LEAD



Multiple Thread Starts

Use the NSTRT modifier to define the number of thread starts. The default is 1, which produces a single start thread. Figure 5-16 shows a two-start multiple thread.

Figure 5-16 NSTRT Example for a Two-Start Thread



Thread Location

Thread location is a function of the dimensional modifiers DEPTH and LENGTH and a START location. Figure 5-17 shows how to locate the thread using the dimensional modifiers DEPTH and LENGTH and a START location.

Digitize the START loc to locate the start of the first cut.

If the digitized loc is not on the MATL entity, CVNC compensates by calculating the correct y-value coincident with that of the MATL entity.

This START loc, shown as point A, is the initial point of the STEPIN vector along which the tool positions to the depth of each threading pass.

The values given to the dimension modifiers DEPTH and LENGTH specify the thread depth and length. DEPTH and LENGTH values determine the location of the initial point of the LEADOUT vector.

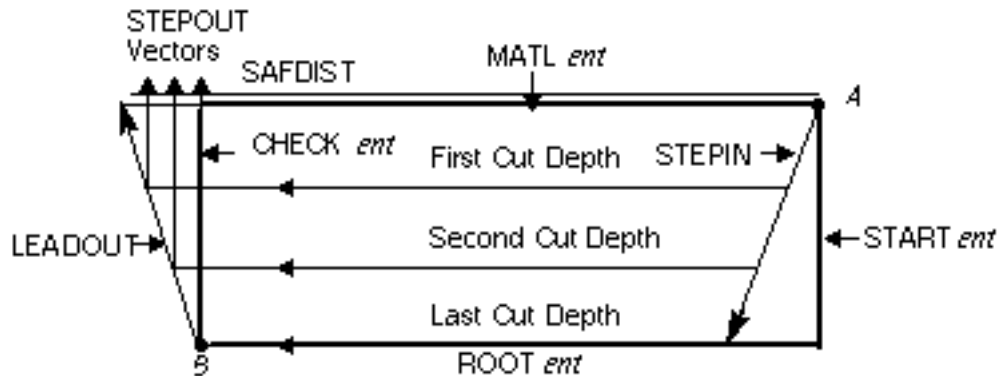
Point B indicates the origin of the LEADOUT vector along which each threading pass is terminated.

Figure 5-17 also shows the STEPOUT vectors along which the tool moves away from the part after the termination of each threading pass.

Please note: Use either the dimensional or the thread location entity modifier sets for tapered threads. The MATL entity slope must match the taper slope.

This figure also shows the STEPOUT vectors along which the tool moves away from the part after the termination of each threading pass.

Figure 5-18 Threading Location Using Entity Modifiers



Please note: Use either the dimensional or the thread location entity modifier sets for tapered threads. The MATL entity slope must match the taper slope.

Threading Passes

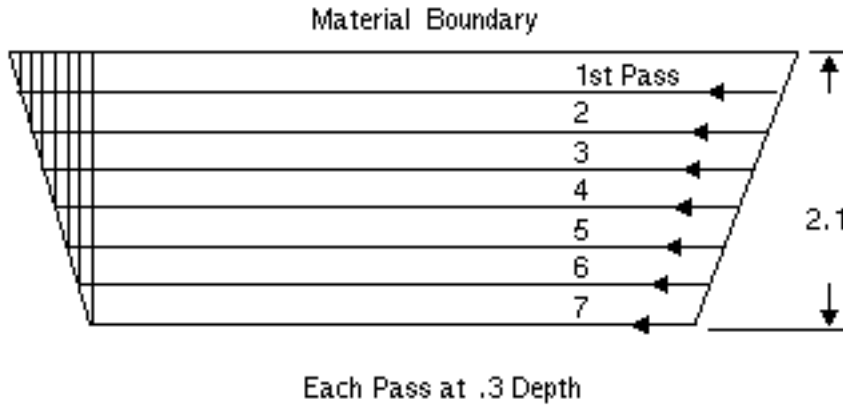
There are three mutually exclusive threading pass modifiers for defining threading pass depths:

- MAXDEP
- NPASS
- FIRST/LAST

MAXDEP specifies a maximum depth allowed for each tool pass. CVNC calculates the necessary number of passes at cut depths equal to or less than MAXDEP. All passes cut equal depths.

Figure 5-19 THREAD Using MAXDEP Modifier

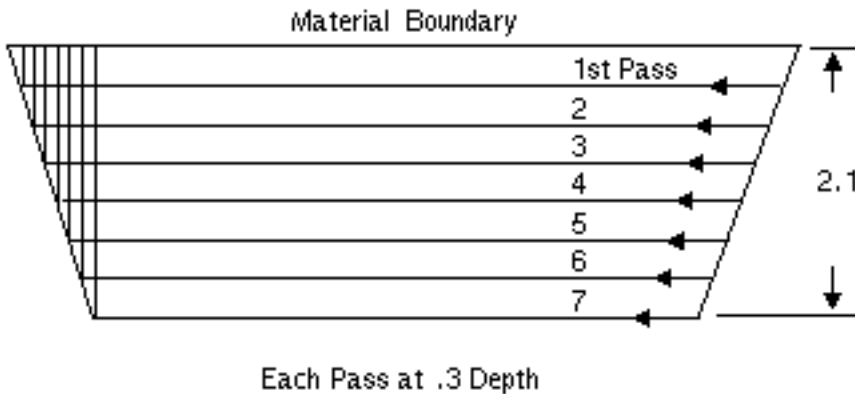
```
NC: > THREAD TPU 12 MATL $L1; DEPTH 2.1 LENGTH 9 START X 4 Y 4 ; MAXDEP  
0.32
```



NPASS specifies the number of tool passes and then CVNC calculates an equal depth for each pass.

Figure 5-20 THREAD Using NPASS Modifier

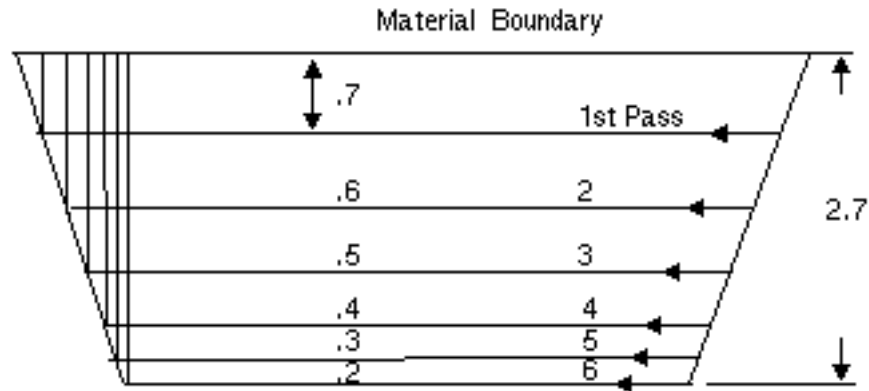
```
NC: > THREAD TPU 12 MATL $L1; DEPTH 2.1 LENGTH 9 START X 4 Y 4 ; NPASS 7
```



FIRST and LAST are used to specify the depth of the first and last tool passes. The FIRST pass must be of a greater depth than the LAST. CVNC calculates linearly decreasing depths for each intermediate pass.

Figure 5-21 THREAD Using FIRST and LAST Modifiers

```
NC: > THREAD TPU 12 MATL $L1; DEPTH 2.1 LENGTH 9 START X 4 Y 4 ; FIRST 0.7
LAST 0.2
```



Threading Into and Out of Material and Gaging Passes

Threading Into and Out of Material

Use the **THRDIN** and **THRDOUT** modifiers for threading into and out of materials. These modifiers create a smooth threaded transition at the beginning and/or ending of the threading passes. These transitions prevent the tool from dwelling at the start or termination of the thread.

- Use **THRDOUT** to include transition out of and motion away from the material while positioning **SAFDIST** away from the material.
- Use **THRDIN** to include a threaded transition into the material while positioning to the depth of each pass.

Gaging Passes

Gaging pass modifiers control repeats of the final threading pass to machine the thread to its finish specification.

The gaging pass modifiers are the following:

- **RPTLST exp** - Specifies the number of times the last pass is repeated. The default is no repeats of the last pass.
- **OPSKIP** - Outputs machine control data necessary for optional skipping of all or remaining last passes.

- **OFFSET exp** - Identifies the new axis offset register to be invoked during repeats of the last pass. Also outputs optional machine stops before and between repeats of the last passes.
- **RETSTRT** - Returns the tool to the starting point (**#CURLOC**) at the end of the threading cycle or between repeats of the last pass.

Hole Processing

CVNC-T2 hole processing commands allow you to perform drilling (DRILL) and tapping (TAP) operations.

Use hole processing commands to do the following:

- Control hole depth
- Control clearance distances
- Identify machining location
- Specify drilling methods and set dwell time

Use the DISPLAY CYCLE command to generate a graphic display of machine tool cycles that result from hole processing operations.

Controlling Hole Depth

You can control hole depth during a DRILL or TAP operation with the DEPTH, THRU, and ENDDIST modifiers.

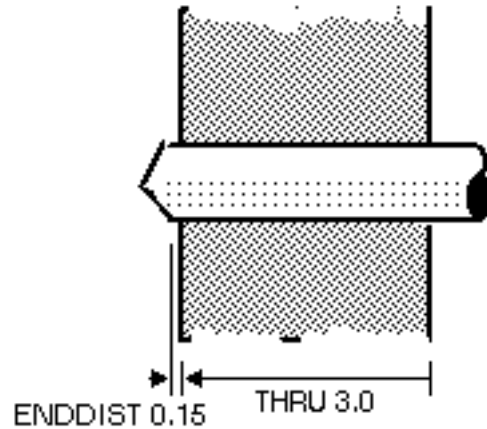
With DRILL and TAP, you can machine to a depth specified with the DEPTH modifier or you can machine a thru-hole with the THRU modifier.

You can modify the DEPTH or THRU value with the ENDDIST modifier. ENDDIST adds the specified value to the input value.

When machining thru-holes, if you do not specify ENDDIST, CVNC adds the current value of the system variable #THRUCLR to the total depth. Use the ENDDIST modifier to set #THRUCLR to a different value.

Figure 5-22 Using the ENDDIST Modifier with THRU

```
NC:>DRILL THRU 3.0 ENDDIST .15  
NC:>SHOW #THRUCLR  
Numeric Value 0.15
```



When machining DEPTH holes, CVNC only adds an ENDDIST value when you use the ENDDIST modifier. This value is stored in the #DEPCLR system variable.

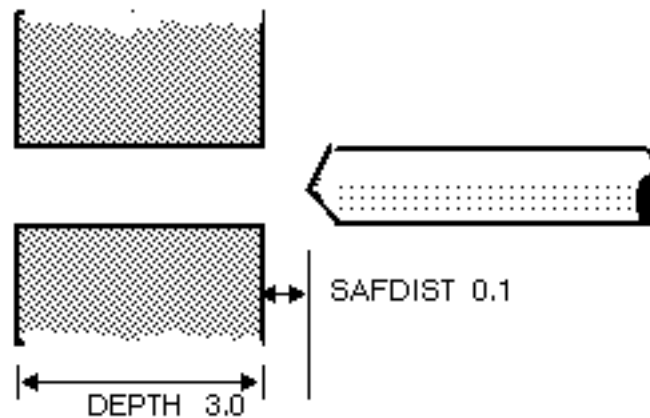
DEPTH and THRU distances apply to the full diameter of the tool. When drilling (with DRILL), however, use the NOANGL modifier so that the tool point angle value is not considered when calculating the drill depth.

Controlling Clearance Distances

Clearance distances are controlled with the SAFDIST modifier.

SAFDIST specifies a value to be added to the x-value of the hole location, as shown in Figure 5-21. This value is used for approaches and retractions.

Figure 5-23 Controlling Clearance with SAFDIST



Identifying the Machining Location

Identify the location of a hole-processing operation with a coordinate location (`loc`) or entities (`ents`) following the `START` modifier, or by specifying the x-coordinate of the starting location with a numeric expression. You can also use `NCGROUPs` to do this.

Machining occurs in the order these locations are entered, unless you do the following:

- Use the `REV` modifier to reverse the order of machining (`locs` or `ents`).
- Specify an `NCGROUP` (in place of `locs` or `ents`) that was defined with the `OPTIM` modifier.

For more information about `NCGROUPs`, see the *CVNC System User Guide and Menu Reference*.

Specifying Drilling Methods

When drilling, choose the default (standard), the `PECK` (deep-hole drilling), or the `BREAK-chip` (pause drilling) method. You can also specify `MAXDEP` and `DWELL` time.

- `PECK` retracts the tool from the hole between passes to the clearance distance.
- `BREAK` causes the tool to pause between passes to the clearance distance.

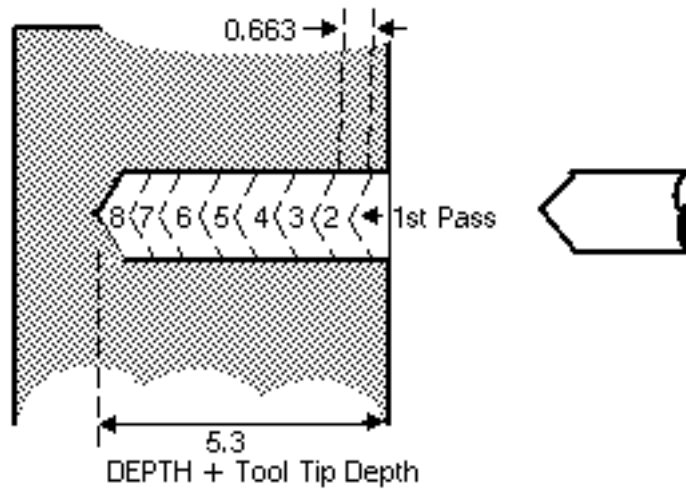
With PECK or BREAK-chip drilling, you must set parameters for determining tool passes.

MAXDEP specifies a maximum depth allowed for each tool pass. CVNC calculates the necessary number of passes at cuts depths equal to or less than MAXDEP. All passes cut equal depths.

Here is a DRILL command with a MAXDEP of 0.75. CVNC calculates that eight tool passes are necessary to cut the specified depth with equal passes that do not exceed MAXDEP. In this example, each pass is 0.663.

Figure 5-24 MAXDEP

NC: >DRILL DEPTH 5.0 PECK MAXDEP 0.75

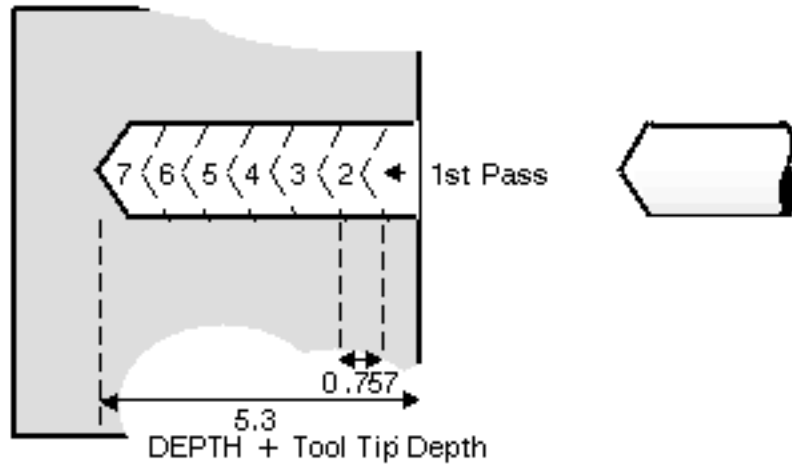


Please note: With DRILL you can specify the amount of dwell time the tool has after it has reached the specified depth. Use DWELL exp to specify dwell time in seconds or REVS exp to specify the number of revolutions of dwell.

NPASS specifies the number of tool passes and then CVNC calculates an equal depth for each pass.

Figure 5-25 NPASS

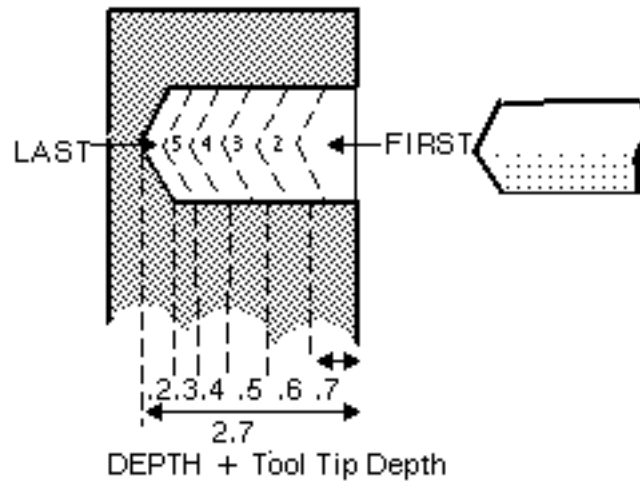
NC: > DRILL DEPTH 5.0 BREAK NPASS 7



FIRST and LAST are used to specify the depth of the first and last tool passes. The FIRST pass must be of a greater depth than the LAST. CVNC calculates linearly decreasing depths for each intermediate pass.

Figure 5-26 Drilling with FIRST and LAST

NC: > DRILL DEPTH 2.4 PECK FIRST 0.7 LAST 0.2



DISPLAY CYCLE Command

The DISPLAY CYCLE command invokes and terminates the graphics emulation of machine tool cycles that result from hole processing commands.

Use `DISPLAY CYCLE ON` to display motion resulting from a hole processing command and for the first location only.

For subsequent locations, only motions to the `APPROACH` and `RETRACT` points are displayed.

`DISPLAY CYCLE OFF` ends cycle emulation for hole processing and subsequently displays only motion to `APPROACH` and `RETRACT` points.

Please note: The `IN` and `INX` commands (see the *CVNC Editor Guide*) cannot be applied to hole processing commands. Hole processing commands cannot be decomposed into equivalent sequences of lower-level `JCF` commands.

Hole Processing for a Tapped Hole

The following `JCF` example shows two hole-processing commands used to machine a tapped hole in the end of a shaft.

Example

```
NC:> DEFTOOL "DRILL" DRILL DIA .119 ATANGL 118
```

```
NC:> DEFTOOL "TAP" DIA .25 TPU 20
```

```
NC:> CHGTOOL 1 "DRILL"
```

```
NC:> DRILL DEPTH 1.0 SAFDIST .1 NOANGL START ENT :Model ent d  
DRILL DEPTH 1.0 SAFDIST .1 NOANGL START ENT $C4 ;
```

```
NC:> MOVE HOME
```

```
NC:> CHGTOOL 2 "TAP"
```

```
NC:> TAP DEPTH .875 SAFDIST .1 START ENT $C4; :Model ent d  
TAP DEPTH .875 SAFDIST .1 START ENT C4;
```

```
NC:> MOVE HOME
```

Additional Motion Commands

Individual motion commands such as CUT and PLUNGE are typically used for specialized finishing operations. These commands allow you to move the cutting tool selectively or interactively along the part being machined. CUT and PLUNGE can also be used to accomplish rough turning.

Individual motion commands are the following:

- **PLUNGE:** Performs motion at the PLUNGE feed rate to the specified location or entities.
- **CUT commands:** Create linear motion by specifying incremental or absolute changes in x- and y-values relative to the current location of the tool. CUT also provides motion at the CUT feed rate TO, ON, or PAST one or two specified entities.
- **CUT ARC:** Performs circular motions along a described arc. The circular motions are based on your specification of the center point, radius, start and end angles, and cutting direction.
- **CUT ENTITY:** Moves the tool along one or more entities. If more than one drive entity is selected, motion is generated sequentially along one entity until the next entity is encountered.

You can intersperse a series of CUT ENTITY commands with changes in stock offset, cutter compensation, or other commands used to specify operational parameters or motion. CUT ENTITY can machine lines, arcs, conics, B-splines, Nsplines, and Cpoles.

CVNC-T2 includes two macros that allow you to easily execute some repetitive finishing operations:

- Use the FTURN macro to perform an entire turning operation, from rough turning through changing stock values to a finish profiling operation. All operations with this macro must use the same tool.
- Use the GROOVE macro to perform multiple cuts to machine grooves on the inside diameter or outside diameter of the part geometry.

Index

A

Accessing
CVNC-T2 2-2

C

CADDS
entities used in CVNC-T2 1-2
CHGAGE (Change Gage Length) command
procedure 4-14
Command files 1-7
configuring the machine 3-2
CPL 4-2
CPL (Construction Plane) command
procedure 4-2
Cplane
changing 4-2
CVNC-T2
command files 1-7
editing data 2-4
entering 2-2
entering data 2-4
features 1-6
macros 1-6
modifying language environment 1-8
output 1-7
pass-through statements 1-7
system monitoring tools 1-6
system variables 1-6

D

DATUM command
procedure 3-2
DEFTOOL (Define Tool) command
procedure 3-7
DISPLAY CYCLE command
procedure 5-23, 5-27
Documentation, printing from Portable
Document Format (PDF) file xii
DRILL command
procedure 5-23

F

Feed rates
definition 4-5
modal 4-5
specifying 4-5
TURN 4-5
Fillet
machining 4-10
Finish turning
definition 5-14
PROFILE 5-14
Finishing operations
CUT 5-29
PLUNGE 5-29

G

Grammar files

definition 1-8

H

Hole processing

clearance 5-24

definition 5-23

depth 5-23

drilling 5-25

graphics emulation 5-28

location for machining 5-25

HOMEPT (Home Point) command

procedure 3-3

J

Job Control File (JCF)

creation 1-5

definition 1-4

example 1-4

modification 2-5

L

Language environment

modification 1-8

M

Machine coordinate system

definition 3-2

Macros 1-6

Motion

commands 5-2

rough turning 5-3

types of 5-2

N

NCGROUP (Entity Grouping) command

with

hole processing 5-25

O

Operation setup 4-1

Operational parameters

tolerances (circular) 4-13

P

Pass-through statements 1-7

Printing documentation from Portable
Document Format (PDF) file xii

PROFILE command

procedure 5-14

Profiling 5-14

PROGRAM command

procedure 2-2

R

Rough turning

contoured motion 5-4

definition 5-3

depth of pass 5-6

noncontoured motion 5-4

TURN 5-3

undercut 5-8

S

Selecting a tool 4-2

Speed rate

specifying 4-5

Spindle speed

specifying 4-5

STOCK (Machining Stock) command

modifiers 4-10

procedure 4-7

T

Tagging entities

definition 2-4

figure 2-3

TAP command

- procedure 5-23
- THREAD command
 - procedure 5-15
- Threading
 - gaging passes 5-21
 - LEADOUT 5-15
 - location 5-17
 - passes 5-19
 - smooth transition 5-21
 - STEPIN 5-15
 - STEPOUT 5-15
 - THREAD 5-15
 - type 5-15
- Tool motion
 - CHGTOOL 3-3
- Tools
 - gage lengths 4-14
- Tools and tool parameters
 - definition 3-7
 - drills 3-7
 - taps 3-7
 - tolerance 4-13
 - turning 3-7
- Twin-turret machine 1-2

